AUTOMOTIVE COMPOSITE DRIVE SHAFTS

DESIGN, COMPUTATIONAL ANALYSIS, AND OPTIMIZATION USING PYTHON

ISBN: 978-81-686144-0-6

Authors:
Dr. Raffi Mohammed
Prof. P. Chakradhar
Dr. P. Raja Sekhar

Publisher
The Institute for Innovations in Engineering and Technology

Automotive Composite Drive Shafts: Design, Computational Analysis, and Optimization Using Python

Dr. Raffi Mohammed
Professor
Department of Mechanical Engineering
Ramachandra College of Engineering (A)
Eluru, Andhra Pradesh, India

Prof. P. Chakradhar
Department of CSE-IoT
Ramachandra College of Engineering (A)
Eluru, Andhra Pradesh, India

Dr. P. Raja Sekhar
Associate Professor
Department of Mathematics
Ramachandra College of Engineering (A)
Eluru, Andhra Pradesh, India

June-2026

Publisher:
The Institute for Innovations in Engineering and Technology
1-102, GP Street, Gurazada,
Pamidimukkala Mandal Krishna (Dt.),
AP-521256,
Website: www.theiiet.com
E-Mail: contact@theiiet.com

ISBN 978-8-16-861440-6

THE INSTITUTE FOR INNOVATIONS IN ENGINEERING AND TECHNOLOGY

Published by **The Institute for Innovations in Engineering and Technology**
1-102, GP Street, Gurazada, Pamidimukkala Mandal, Krishna (Dt.), Andhra Pradesh-521256.

Title of the Book: **Automotive Composite Drive Shafts: Design, Computational Analysis, and Optimization Using Python**, *June, Copyright © 2026 with Authors.*

Authors:

Dr. Raffi Mohammed, Professor, Department of Mechanical Engineering, Ramachandra College of Engineering(A), Eluru, Andhra Pradesh, India
Prof. P. Chakradhar, Department of CSE-IoT, Ramachandra College of Engineering(A), Eluru, Andhra Pradesh, India
Dr. P. Raja Sekhar, Department of Mathematics**,** Ramachandra College of Engineering (A)**,** Eluru, Andhra Pradesh, India

No part of this publication may be reproduced or distributed in any form or by any means, electronic, mechanical, photocopying, recording or otherwise or stored in a database or retrieval system without the prior written permission of the publisher or editors. The program listings (if any) may be entered, stored and executed in a computer system, but they may not be reproduced for publication.

This edition can be exported from India only by the publishers,
The Institute for Innovations in Engineering and Technology

Information contained in this work has been obtained by The Institute for Innovations in Engineering and Technology, from sources believed to be reliable. However, neither The Institute for Innovations in Engineering and Technology nor its authors guarantee the accuracy or completeness of any information published herein, and neither The Institute for Innovations in Engineering and Technology (India) nor its authors shall be responsible for any errors, omissions, or damages arising out of use of this information. This work is published with the understanding that The Institute for Innovations in Engineering and Technology and its authors are supplying information but are not attempting to render engineering or other professional services. If such services are required, the assistance of an appropriate professional should be sought.

ISBN 978-8-16-861440-6

Typeset at the IIET, D: 1-102, GP Street, Vijayawada-521256. Printed and bounded in India at Printster.in, S-548A, 1st Floor, School Block, Shakarpur, Laxmi Nagar, Delhi, 110092, India
Visit us at: www.theiiet.com ; Phone: 91-9533111789;
Write to us at: contact@theiiet.com

Acknowledgements

The authors express their profound gratitude to **Dr. M. Muralidhara Rao**, Director & Principal, Ramachandra College of Engineering (Autonomous), Eluru, for his exceptional leadership, continuous encouragement, and unwavering support throughout the development of this book. His commitment to fostering a vibrant research culture, promoting innovation, and enabling knowledge creation has been a constant source of inspiration during the preparation of this work.

The authors extend their sincere thanks to **Dr. S. Subramanya Sarma**, Dean of Research, Ramachandra College of Engineering (Autonomous), Eluru, for his academic stewardship, insightful guidance, and valuable encouragement. His dedication to interdisciplinary research, academic excellence, and scholarly advancement has significantly enriched the quality and depth of this publication.

The authors wish to place on record their heartfelt appreciation to **Mr. K. Venugopal, Chairman**, and **Mr. K. Sai Rohith, Managing Director & Secretary, Ramachandra College of Engineering (Autonomous), Eluru**, for their visionary leadership, constant encouragement, and unwavering support. Their commitment to research excellence, innovation, and institutional development has created an environment that nurtures intellectual growth and scholarly contributions.

The authors also express their sincere gratitude to the faculty members, researchers, and academic colleagues of Ramachandra College of Engineering (Autonomous), Eluru, particularly from the Department of Mechanical Engineering and allied disciplines, whose valuable discussions, technical insights, and cooperation contributed significantly to the successful completion of this book.

Special appreciation is extended to the scientific and engineering communities worldwide whose pioneering research in composite materials, computational modeling, finite element analysis, optimization techniques, and automotive engineering has provided the foundation upon which this work is built. Their relentless pursuit of innovation continues to inspire advancements in lightweight, high-performance engineering systems.

The authors are deeply grateful to all researchers, academicians, industry professionals, and students whose interest in advanced composite structures, computational design, and sustainable automotive technologies motivates the dissemination of knowledge through publications such as this.

Finally, the authors dedicate this book to the readers, scholars, engineers, educators, and students who strive to explore emerging technologies in automotive engineering. It is hoped that this volume will serve as a valuable resource for advancing research, innovation, and practical applications in the field of composite drive shaft design and optimization.

Authors

Preface

The automotive industry is undergoing a transformative shift toward lightweight, high-performance, and sustainable engineering solutions. Among the critical components influencing vehicle efficiency, performance, and fuel economy, the drive shaft plays a vital role in transmitting power from the engine to the wheels. Traditional metallic drive shafts, while reliable, are often associated with increased weight, lower specific strength, and limitations in vibration damping. Composite materials have emerged as a promising alternative, offering superior strength-to-weight ratios, enhanced fatigue resistance, improved corrosion resistance, and greater design flexibility.

This book, *Automotive Composite Drive Shafts: Design, Computational Analysis, and Optimization Using Python*, presents a comprehensive exploration of composite drive shaft technology, integrating fundamental concepts, computational modeling, finite element analysis, and modern optimization approaches. The work emphasizes the application of advanced composite materials such as carbon fiber, glass fiber, Kevlar, and hybrid composites in the design of automotive drive shafts to achieve significant weight reduction while maintaining structural integrity and performance requirements.

A distinctive feature of this book is the incorporation of Python-based computational techniques for engineering analysis and optimization. Python has emerged as a powerful tool for engineers and researchers due to its versatility, extensive scientific libraries, and ability to automate complex calculations. By combining traditional engineering principles with modern computational methods, this book demonstrates how data-driven decision-making and optimization techniques can be effectively utilized to enhance drive shaft design and material selection.

The chapters systematically cover the fundamentals of composite materials, drive shaft design considerations, computer-aided modeling, finite element analysis, vibration and stress evaluation, material selection methodologies, optimization techniques, and practical case studies. The content has been developed to bridge the gap between theoretical concepts and industrial applications, making it valuable for students, researchers, academicians, and practicing engineers.

This book is intended to serve as a reference for undergraduate and postgraduate students, faculty members, research scholars, automotive engineers, and professionals involved in the fields of composite materials, mechanical design, computational engineering, and vehicle technology. It is our sincere hope that this publication will contribute to the advancement of research and innovation in lightweight automotive structures and inspire future developments in sustainable transportation systems.

The authors acknowledge that engineering knowledge continuously evolves with technological advancements. Therefore, constructive suggestions and feedback from readers are warmly welcomed and will be invaluable for future editions.

We sincerely hope that this book provides meaningful insights and practical guidance to its readers and contributes to the growing body of knowledge in automotive composite engineering.

Dr. Raffi Mohammed
Prof. P. Chakradhar
Dr. P. Raja Sekhar

Foreword

The pursuit of lightweight, efficient, and sustainable engineering solutions has become a defining objective of modern automotive design. As global industries strive to enhance vehicle performance while reducing energy consumption and environmental impact, advanced composite materials have emerged as a transformative technology with immense potential. Among the many automotive components benefiting from these advancements, the drive shaft represents a critical system where material innovation can significantly improve performance, durability, and efficiency.

The book *Automotive Composite Drive Shafts: Design, Computational Analysis, and Optimization Using Python* presents a timely and comprehensive contribution to this evolving field. By integrating the principles of composite materials, computational engineering, finite element analysis, and Python-based optimization techniques, the authors have developed a valuable resource that bridges the gap between theoretical understanding and practical engineering applications.

One of the notable strengths of this book is its multidisciplinary approach. The authors effectively combine materials science, mechanical design, computational modeling, and data-driven optimization methodologies to address contemporary challenges in automotive engineering. The inclusion of Python as a computational platform reflects the growing importance of programming and automation in engineering analysis, enabling researchers and practitioners to perform efficient design evaluations, simulations, and optimization studies.

The contents of this book are presented in a systematic and reader-friendly manner, covering the fundamentals of composite materials, drive shaft design considerations, modeling approaches, structural and vibration analysis, optimization strategies, and practical case studies. Such comprehensive coverage makes this volume highly relevant not only for students and researchers but also for engineers and industry professionals seeking innovative solutions in lightweight structural design.

The authors deserve appreciation for their dedication in compiling this work and for their efforts in promoting the application of modern computational tools in mechanical and automotive engineering. Their contribution aligns with the increasing demand for sustainable engineering practices and intelligent design methodologies that will shape the future of transportation systems.

I am confident that this book will serve as an important reference for academicians, researchers, engineers, and students engaged in the fields of composite materials, automotive engineering, computational mechanics, and design optimization. It is my sincere hope that the knowledge presented herein will inspire further research, innovation, and technological advancement in the development of next-generation automotive components.

I extend my best wishes to the authors for the success of this publication and trust that it will make a meaningful contribution to engineering education, research, and industrial practice.

Dr. Abdul Siddique Shaik
Faculty of Mechanical Engineering
King Khalid University
Abha, Saudi Arabia

CONTENTS **PAGE NO.**

CHAPTER-I

INTRODUCTION

1.1 INTRODUCTION 1

1.2 DESCRIPTION OF THE PROBLEM 1

1.3 AIM AND SCOPE OF THE WORK 1

1.4 UNIVERSAL JOINT 2

1.5 PURPOSE OF THE DRIVE SHAFT (OR PROPELLER SHAFT) 4

1.6 PROPELLER SHAFT ARRANGEMENT IN AUTOMOBILES 4

1.7 SPECFICATIONS OF TAYOTA QUALIS 6

1.8 DEMERITS OF CONVENTIONAL DRIVE SHAFT 7

CHAPTER-II

COMPOSITE MATERIALS

2.1 COMPOSITE MATERIALS 8

2.2 CLASSIFICATION OF COMPOSITE MATERIALS 8

2.3 PROPERTIES OF COMPOSITE MATERIALS 10

2.4 ADVANTAGES OF COMPOSITES OVER THE CONVENTIONAL 11

2.5 LIMITATIONS OF COMPOSITES 12

2.6 APPLICATIONS OF COMPOSITES 12

2.7 MERITS OF COMPOSITE DRIVE SHAFT 13

2.8 MANUFACTURING OF COMPOSITE DRIVE SHAFT 14

CHAPTER-III

3 LITERATURE SURVEY 15

CHAPTER-IV

CATIA

4.1 INTRODUCTION 17

4.2 BASIC WORKBENCHES IN CATIA V5 17

4.3 SALIENT FEATURES OF CATIA V5-R16 19

CHAPTER-V

MODELLING OF PROPELLER SHAFT ASSEMBLY USING CATIA V5-R16

5.1 MODELLING OF UNIVESRAL JOINT 20

5.2 MODELING OF CENTRE BLOCK 24

5.3 MODELLING OF PROPELLER SHAFT ARRANGEMENT 26

5.4 MODELLING OF SLIP YOKE 28

5.5 ASSEMBLY OF PROPELLER SHAFT ARRANGEMENT 32

CHAPTER-VI

FINITE ELEMENT METHODS

6.1 INTRODUCTION 33

6.2 GENRAL PROCEDURE OF FEA 35

6.3 THE PROCESS OF F.E.A 36

6.4 ADVANTAGES OF FEM 37

6.5 LIMITATIONS OF FEM 38

6.6 APPLICATIONS OF FINITE ELEMENT METHODS 39

6.7 BASIC APPROACH TO FEA 40

6.8 POPULAR FEA SOFTWARES 43

CHAPTER-VII

ANSYS

7.1 INTRODUCTION 44

7.2 SPECIFIC CAPABILITIES OF ANSYS 44

7.3 STRUCTURE OF ANSYS 44

7.4 STEP BY STEP PROCESSING OF GOOD ANALYSIS 45

7.5 ADVANTAGES OF ANSYS 46

CHAPTER-VIII

ANALYSIS ON PROPELLER SHAFT ASSEMBLY OF TOYOTA QUALIS

8.1 ANSYS WORKBENCH 47

8.2 ANALYSIS PROCEDURE 48

8.3 MATERIALS USED IN THE ANALYSIS AND THEIR PROPERTIES 50

8.4 WEIGHT CALCULATIONS 52

8.5 MESHING OF ASSEMBLY 53

8.6 REPORT ON STRUCTURAL STEEL 54

8.7 REPORT ON E GLASS 71

8.8 REPORT ON E CARBON 75

8.9 REPORT ON E GLASS POLYESTER RESIN 79

CHAPTER-IX

Implementation of Python Programming for Validation of Design and Finite Element Analysis Results of Automotive Composite Drive Shafts 84-90

CHAPTER-X

CONCLUSION 91

BIBILOGRAPHY **92**

CHAPTER-1

1.1 INTRODUCTION

The advanced composite materials such as Graphite, Carbon, Kevlar and Glass with suitable resins are widely used because of their high specific strength (strength/density) and high specific modulus (modulus/density). Advanced composite materials seem ideally suited for long, power driver shaft (propeller shaft) applications. Their elastic properties can be tailored To increase the torque they can carry as well as the rotational speed at which they operate. The drive shafts are used in automotive, aircraft and aerospace applications. The automotive industry is exploiting composite material technology for structural components construction in order to obtain the reduction of the weight without decrease in vehicle quality and Reliability. It is known that energy conservation is one of the most important objectives in vehicle design and reduction of weight is one of the most effective measures to obtain this result. Actually, there is almost a direct proportionality between the weight of a vehicle and its fuel consumption, particularly in city driving.

1.2 DESCRIPTION OF THE PROBLEM

Almost all automobiles (at least those which correspond to design with rear wheel drive and front engine installation) have transmission shafts. The weight reduction of the drive shaft can have a certain role in the general weight reduction of the vehicle and is a highly desirable goal, if it can be achieved without increase in cost and decrease in quality and reliability. It is possible to achieve design of composite drive shaft with less weight to increase the first natural frequency of the shaft and to decrease the bending stresses using various stacking sequences. By doing the same, maximize the torque transmission and torsional buckling capabilities are also maximized.

1.3 AIM AND SCOPE OF THE WORK

This work deals with the replacement of a conventional steel drive shaft with E-GLASS/ Epoxy, High Strength E-CARBON/Epoxy and E-GLASS POLYESTER composite drive shafts for an automobile application.

1.4 UNIVERSAL JOINT

A universal joint is a particular type of connection between two shafts, whose axes are inclined to each other. The simplest type of universal joint is the Hooke's joint which is most widely used because of the fact that it is simple and compact in construction and reasonably efficient at small angles of propeller shaft movement up and down, say up to 18 degrees. The axes of shafts A and B are intersecting. Each of these shafts contains a yoke. The cross C has four arms. The two opposite arms of the cross are supported in bushes in the yoke of shaft A, while the other two arms of the cross are supported in the yoke of shaft B. Thus shaft A can have angular rotation about the axis XX and the shaft B, about the axis YY. It is thus seen that it will be possible with the Hooke's joint for the shafts A and B to have positive drive while allowing angular movement between them.

An improved form of the Hooke's joint uses needle roller bearings to support the cross in the yokes. This results in increase of joint efficiency. A perfect circle U-joint which has special feature in that bearings races on the inside are crowned, which minimizes galling and flaking by distributing load evenly.

In a flexible ring universal joint each shaft carries a three-arm spider on splines. There are six holes in the flexible ring which is made of reinforced rubberized fabric. Each of the spiders is fixed to each side of the ring by means of bolts and nuts. This type of joint is thus very simple in construction and hence cheap. There is also need for lubrication of the joint. It also provides small axial movement. The only disadvantage is that it cannot operate at large angular deflections. Further, to transmit large amount of torque to size of the joint becomes unduly large.

The universal joints described above have one defect common. In all these joints, the speed of the driven shaft does not remain uniform. Depending upon the angle of inclination of the shafts, the driven shaft sped undergoes cyclic variation. This variation is zero for zero angle of inclination, but its magnitude becomes considerable when the angle is large.

It must be appreciated that in case of hook's joint with the needle roller bearings, it would be desirable to have a small operating angle between the shafts than to have none at all, because in the former case, the needle would roll slightly, thus preventing the high stress contact areas remaining at same place continuously thus avoiding the squeezing of grease and consequently preventing their embedding into the journal and the cap race.

One method to achieve a uniform driven shaft speed is by using two such joints. The intermediate shaft is so arranged that it makes equal angles theta with the first and third shafts. The variation caused by one joint is cancelled out by the second joint. However, this will be valid only when the angles on the both joints are exactly equal, which is not always the case in practice. Special constant velocity universal joints where the fluctuations in the sped of the driven shaft at very large angles are completely absent are also available though these are much costlier and complicated in construction. These types of joints have to be used where due to location of the engine close to the wheels; the connecting shafts are short; for example, in the case of the front wheel drive, in the four wheel drive vehicles etc.

In the front wheel drive, the engine torque has to be transmitted through members that rise and fall due to road shocks and also turn from side to side while steering the vehicle. Moreover, the shafts must be able to slide in and out as large operating angles are involved, the shaft being of smaller length. Basically there are two types of constant velocity joints, the fixed type and plunging type. The fixed type or the outboard type joint is employed on the wheel end of the drive shaft while the plunging type or the inboard type is used on the differential end of drive shaft.

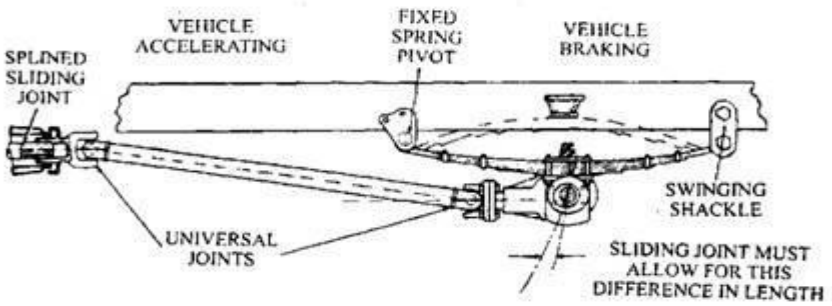
The first real constant velocity joint, still in use, is the Rzeppa joint. In this six spherical balls are held in a precise geometric position midway between the two shafts, bisecting the angle between them. Almost concurrently with the Rzeppa joint, another constant velocity joint developed in France was the tripod joint, which used three roller bearings attached to arms at the end of the driving shaft. A further modification of the original design was the constant velocity joint with plunging capability. This permits the driving and the driven shafts to move toward or away from each other. Three types of such joints are there. Out of these the closed tulip type and the open tulip type are basically the tripod configurations.

A number of prefect circle constant velocity joints, both of the outboard or fixed type and the inboard or plunging type. The outboard joint is used at the wheel end in case of front wheel drive vehicle, while the inboard joint is located on each shaft at the differential end and allows the slipping motion required for change in the length of the drive shaft in response to suspension system action when the vehicle is traveling over irregular surface.

1.5 PURPOSE OF THE DRIVE SHAFT (OR PROPELLER SHAFT)

- It must transmit torque from the transmission to the differential gear box
- The drive shaft must also be capable of rotating at the very fast speed required by the vehicle.
- The drives shaft must also operate through constantly changing the angles between the transmission, the differential and the axels.
- The length of the drive shaft must also be capable of changing while transmitting torque.

1.6 PROPELLER SHAFT ARRANGEMENT IN AUTOMOBILES



This is a shaft which transmits the drive from the transmission to the bevel pinion or worm of final drive in the front engine, rear drive vehicles and from the transfer box to the front and rear axles in all-wheel drive vehicle. It is also called drive shaft. It mainly consists of three parts

- Shaft- As this has to withstand mainly torsion loads, it is usually made of tubular cross-section. It also has to be well balanced to avoid whirling at high speeds. Shafts are made of steel, aluminum or composite materials.

- One or two universal joints, depending upon the type of rear axle drive used. The universal joints account for the up and down movements of the rear axle when the vehicle is running. Modern vehicles use, however, cardan joints or high-speed constant velocity joints, double cardan joints or rubber couplings with options for intermediate bearings, limited slip devices and crash features that absorb energy.

- Slip joint-Depending upon the type of drive, one slip joint may be there in shaft. This serves to adjust the length of the propeller shaft when demanded by the rear axle movements.

A propeller shaft consists of two universal joints at the ends and a slip or sliding joint. Slip joint is formed by the internal splines on the sleeve attached to the left universal joint and external splines on the propeller shafts.

In some designs, slip arrangement is slightly different. In these a universal joint and slip yoke are located at the transmission end of the shaft where these are held in alignment by a bushing in the transmission rear extension. This spline is lubricated internally by transmission lubrication or grease. One such design is propeller shaft with solid tube. Sometimes a rubber element is incorporated in-between the two sliding tubes to make the relative movement smooth and noiseless.

In vehicles with large wheel base, the long propeller shaft would tend to sag and whirl is like the action of a rope that is in arc while held at both ends. At a certain speed the whirling becomes critical and shaft vibrates violently. This also sets up sympathetic resonant vibrations in the vehicle body. Critical whirling speed of shafts can be increased by increasing its diameter, but that would increase its inertia which would decrease its acceleration and deceleration. Critical whirling speed is also found to decrease as the square of its length. Thus decreasing the length to half would increase the critical speed four times. In some designs this has been achieved by extending the rear end of the transmission main shaft and housing while in others, by extending the final drive pinion shaft and housing.

Another method to decrease the shaft length is to use divided propeller shaft, supported by intermediate bearings. Other advantages of such arrangement are the lower floor height and possibility of achieving large offsets between transmission centre line and the final-drive pinion center line in commercial vehicles in two or more stages. An example is a two-piece propeller shaft used in Ashok Leyland vehicles in India. It consists of two propeller shafts supported in the middle by a self-aligning ball bearing fitted in cross member of chassis frame. In all the there are 3 universal joints and 2 slip joints. At the end there are flange yokes which fitted to the gear box shaft and the rear axle pinion shafts.

1.7 SPECIFICATIONS OF TAYOTA QUALIS

The Qualis is powered by a mighty, high-performance 2.4-litre diesel engine. Delivering maximum power of 54 kW at 4200 rpm and maximum torque of 151 Nm at 2400 rpm, the 2.4 liter straight four-cylinder diesel engine guarantees smooth starts and powerful acceleration with a maximum speed of 130 km/h.

Specifications

- **Dimensions**

Length	4425 mm
Width	1655 mm
Height	1880 mm
Kerb Weight	1505 Kg
Ground Clearance	178 mm
Wheel Base	2500 mm

- **Power**

Engine Type	4-cylinder In-line, 8-valve, OHC Belt Drive
Piston Displacement	2446 cc
Max. Power	75 PS @ 4200 rpm
Max. Torque	15.4 Kgm @ 2400 rpm

- **Transmission & Gear Box Type** 5M/T

- **Suspension**

Front	Double wishbone with Torsion Bar
Rear	Leaf Spring, Rigid

- **Brakes**

  Type - Superior Anti-fade Braking System with Load Sensing & proportioning valve which adjusts the braking performance as per axle loading.

  Front - Ventilated Disc

  Rear - Drum

  Supplementary Rear brake LSPV & BV

- **Tyres** 175R 14C

- **Fuel Tank Capacity** 53 Liters

1.8 DEMERITS OF A CONVENTIONAL DRIVE SHAFT

- They have less specific modulus and strength
- Increased weight
- Conventional steel drive shafts are usually manufactured in two pieces to increase the fundamental bending natural frequency because the bending natural frequency of a shaft is inversely proportional to the square of beam length and proportional to the square root of specific modulus. Therefore the steel drive shaft is made in two sections connected by a support structure, bearings and U-joints and hence overall weight of assembly will be more.
- Its corrosion resistance is less as compared with composite materials.
- Steel drive shafts have less damping capacity.

CHAPTER-2: COMPOSITES

2.1 COMPOSITE MATERIALS

Composites consist of two or more materials or material phases that are combined to produce a material that has superior properties to those of its individual constituents. The constituents are combined at a macroscopic level and or not soluble in each other. The main difference between composites, where as in alloys, constituent materials are soluble in each other and form a new material which has different properties from their constituents.

The advanced composite materials such as graphite, carbon, Kevlar and Glass with suitable resins are widely used because of their high specific strength (strength/density) and high specific modulus (modulus/density). Advanced composite materials seem ideally suited for long, power driver shaft (propeller shaft) applications. Their elastic properties can be tailored to increase the torque they can carry as well as the rotational speed at which they operate. The drive shafts are used in automotive, aircraft and aerospace applications. The automotive industry is exploiting composite material technology for structural components construction in order to obtain the reduction of the weight without decrease in vehicle quality and reliability. It is known that energy conservation is one of the most important objectives in vehicle design and reduction of weight is one of the most effective measures to obtain this result. Actually, there is almost a direct proportionality between the weight of a vehicle and its fuel consumption, particularly in city driving.

2.2 CLASSIFICATION OF COMPOSITE MATERIALS

Composite materials are commonly classified at following two distinct levels:

First level of Classification is based on matrix constituent

- **Polymer matrix composites**

Polymers make ideal materials as they can be processed easily; possess lightweight and desirable mechanical properties. Two main kinds of polymers are Thermosets and Thermoplastics Thermosets have qualities such as a well-bonded three dimensional molecular structure after curing they decompose instead of melting on Hardening. Thermoplastics have one- or two-dimensional molecular structure and they tend to melt at an

elevated temperature and show exaggerated melting point. Another advantage is that the process of softening at elevated temperatures can reversed to regain its properties during cooling.

- **Metal matrix composites**

Metal matrix composites, at present though generating a wide interest in research fraternity, are not as widely in use as their plastic counterparts. High strength, Fracture toughness and stiffness are offered by metal matrices than those offered by their polymer counterparts. They can withstand elevated temperature in corrosive environment than polymer composites. Most metals and alloys could be used as matrices and they require reinforcement materials which need to be stable over a range of temperature and non-reactive too.

- **Ceramic Matrix composites**

Ceramics can be described as solid materials which exhibit very strong Ionic bonding in general and in few cases covalent bonding. High melting points, good corrosion resistance, stability at elevated temperatures and high compressive strength, render ceramic-based matrix materials a favorite for applications requiring a structural material that doesn't give way at temperatures above 1500ºC.Naturally, ceramic matrices are the obvious choice for high temperature applications.

Second level of classification is based on reinforcement form.

- **Fiber reinforced composites**

Fiber reinforced composites are composed of fibers embedded in matrix material. Such a composite is considered to be a discontinuous fiber or short fiber composite if its properties vary with fiber length. On the other hand, when the length of the fiber is such that any further increase in length does not further increase, the elastic modulus of the composite, the composite is considered to be continuous fiber reinforced composites

- **Laminar reinforced composites**

Laminar reinforced composites are composed of layers of materials held together by matrix. Sandwich structures fall under this category.

- **Particulate reinforced composites**

Particulate reinforced composites are composed of particles distributed or embedded in a matrix body. The particles may be flakes or in powder form. Concrete and wood particle boards are examples of this category.

Technologically, the most important composites are those in which the dispersed phase is in the form of a fiber. The Design of fiber-reinforced composites is based on the specific strength. Specific strength is the ratio between strength and density. Specific modulus is the ratio between modulus and density. Fiber length has a great influence on the mechanical characteristics of a material. The fibers can be either long or short. Long continuous fibers are easy to orient and process, while short fibers cannot be controlled fully for proper orientation. Long fibers provide many benefits over short fibers. These include impact resistant, low shrinkage, improved surface finish and dimensional stability. However short fiber provide low cost are easy to work with and have fast cycle time fabrication procedures.

The principal fibers in commercial use are various types of glass, carbon, graphite, Kevlar. All these fibers can be incorporated into a matrix either in continuous lengths or in discontinuous lengths as shown in the Fig. The matrix material may be a plastic or rubber polymer, metal or ceramic. Laminate is obtained by stacking a number of thin layers of fibers and matrix consolidating them to the desired thickness. Fiber orientation in each layer can be controlled to generate a wide range of physical and mechanical properties for the composite laminate.

2.3 PROPERTIES OF COMPOSITE MATERIALS

The physical properties of composite materials are generally not isotropic (independent of direction of applied force or load) in nature, but rather are typically orthotropic (depends on the direction of the applied force or load). For instance, the stiffness of a composite panel will often depend upon the orientation of the applied forces and/or moments. Panel stiffness is also dependent on the design of the panel.

In contrast, isotropic materials (for example, aluminum or steel), in standard wrought forms, typically have the same stiffness regardless of the directional orientation of the applied forces and/or moments. While, composite materials exhibit different properties in different directions.

The relationship between forces/moments and strains/curvatures for an isotropic material can be described with the following material properties: Young's Modulus, the Shear Modulus and the Poisson's ratio, in relatively simple mathematical relationships. For the anisotropic material, it requires the mathematics of a second order tensor and up to 21 material property constants. For the special case of orthogonal isotropy, there are three different material property constants for each of Young's Modulus, Shear Modulus and Poisson's ratio--a total of 9 constants to describe the relationship between forces/moments and strains/curvatures.

2.4 ADVANTAGES OF COMPOSITES OVER THE CONVENTIONAL MATERIALS

- High strength to weight ratio
- High stiffness to weight ratio
- High impact resistance
- Better fatigue resistance
- Improved corrosion resistance
- Good thermal conductivity
- Low coefficient of thermal expansion. As a result, composite structures may exhibit a better dimensional stability over a wide temperature range.
- High damping capacity.

2.5 LIMITATIONS OF COMPOSITES

- Mechanical characterization of a composite structure is more complex than that of metallic structure
- The design of fiber reinforced structure is difficult compared to a metallic structure, mainly due to the difference in properties in directions
- The fabrication cost of composites is high
- Rework and repairing are difficult
- They do not have a high combination of strength and fracture toughness as compared to metals
- They do not necessarily give higher performance in all properties used for material selection.

2.6 APPLICATIONS OF COMPOSITES

The common applications of composites are extending day by day. Nowadays they are used in medical applications too. The other fields of applications are,

- Automotive : Drive shafts, clutch plates, engine blocks, push rods, frames, Valve guides, automotive racing brakes, filament–wound fuel tanks, fiber Glass/Epoxy leaf springs for heavy trucks and trailers, rocker arm covers, suspension arms and bearings for steering system, bumpers, body panels and doors
- Aircraft: Drive shafts, rudders, elevators, bearings, landing gear doors, panels and floorings of airplanes etc.
- Space: payload bay doors, remote manipulator arm, high gain antenna, antenna ribs and struts etc.
- Marine: Propeller vanes, fans & blowers, gear cases, valves &strainers, condenser shells.

- Chemical Industries: Composite vessels for liquid natural gas for alternative fuel vehicle, racked bottles for fire service, mountain climbing, underground storage tanks, ducts and stacks etc.

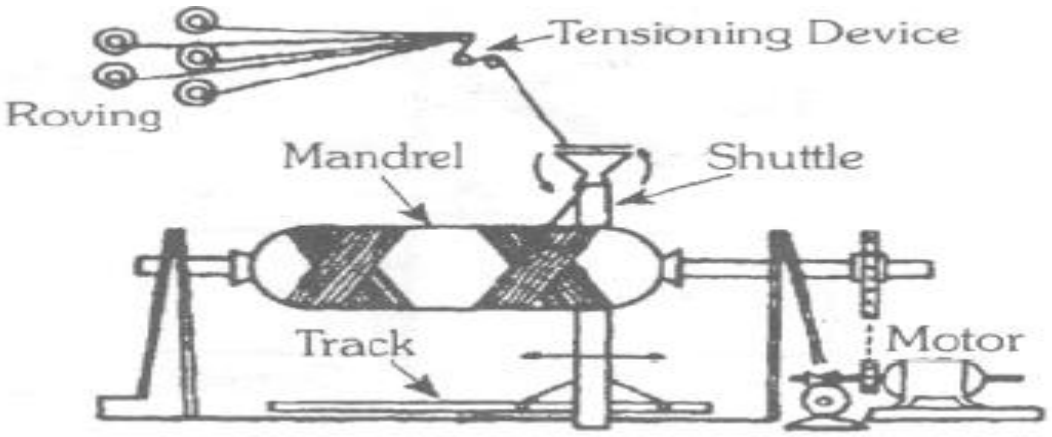
- Electrical & Electronics: Structures for overhead transmission lines for railways, Power line insulators, Lighting poles, Fiber optics tensile members etc.

2.7 MERITS OF COMPOSITE DRIVE SHAFT

- They have high specific modulus and strength
- Reduced weight
- The fundamental natural frequency of the carbon fiber composite drive shaft can be twice as high as that of the steel or aluminum because the carbon fiber composite material has more than 4times the specific stiffness of, which makes it possible to manufacture the drive shaft of passenger cars in one piece. A one-piece composite shaft can be manufactures so as to satisfy the vibration requirements. This eliminates all the assembly, connecting the two piece steel shafts and thus minimizes the overall weight, vibrations and cost.
- Due to weight reduction, fuel consumption will be reduced.
- They have high damping capacity and hence they produce less vibration and noise.
- They have good corrosion resistance
- Greater torque capacity than steel and aluminum shaft
- Longer fatigue life than steel and aluminum shaft

2.8 MANUFACTURING OF COMPOSITE DRIVE SHAFT

Filament winding process is used in the fabrication of composite drive shafts. It is automated processes for creating parts of simple geometry wherein continuous resin impregnated fibers are wound over a rotating male tool called mandrel. Fig 10.3 depicts a filament winding process where a continuous fiber roving passes through a shuttle, which rotates and the roving is wrapped around a revolving or stationary mandrel



After completion of the winding, the filament wound structure is cured at room temperature or in an oven. The mandrel is removed after the curing. The mandrel, which determines accurate internal geometry for the component, is generally the only major tool. Low cost mandrel materials such as cardboard or wood can be used for winding low cost routine parts. For critical parts requiring close tolerances, expensive mandrels designed for long term use may be required. For high temperature cure 315°C (600°F), graphite mandrels with low thermal expansion may be advantageous. However, attention should be paid for potential difficulties for mandrel removal.

In this technique the winding angle, fiber tension, and resin content can be varied. Filament winding is relatively inexpensive, repetitive and accurate in fiber placement.

CHAPTER-3: LITERATURE SURVEY

This project was carried out by referring International Journal of Engineering and Technology, Vol. 3, No.2, 2006, pp. 227-237. In this journal, authors had worked out the design optimization of composite drive shafts transmitting very large torques.

With reference to this journal I opted to do my project on material optimization on propeller shaft of Toyota Quails which transmits a maximum torque of 154N-m at 2400 rpm. I also extended my project by including more composite materials. The composites selected for this analysis are – carbon Epoxy, Glass Epoxy and E Glass Polyester Resin along with structural steel. The brief abstract of the above stated journal is discussed below.

Drive shafts as power transmission tubing are used in many applications, including cooling towers, pumping sets, aerospace, trucks and automobiles. In metallic shaft design, knowing the torque and the allowable shear stress for the material, the size of the shaft's cross section can be determined. As the geometric parameter (polar moment of inertia of the cross-sectional area divided by the outer radius) equal to the torque divided by the allowable shear stress, there is unique value for the shaft inner radius when the outer radius is limited by the space under the car cabin. Metallic drive shaft has the limitations of weight, low critical speed and vibrational characteristics. Composite drive shafts have solved many automotive and industrial problems accompany the usage of the conventional metal ones because the performance is limited due to lower critical speed, weight, fatigue and vibration. Numerous solutions such as flywheels, harmonic dampers, vibration shock absorbers and multiple shafts with bearings, couplings, and heavy associated hardware have shown limited success in overcoming the problems.

When the length of steel drive shaft is beyond 1500 mm, it is manufactured in two pieces to increase the fundamental natural frequency, which is inversely proportional to the square length and proportional to the square root of specific modulus. The nature of composites with their higher specific modulus (modulus to density), which in carbon/epoxy exceed four times that of aluminum, enables the replacement of the two pieces metal shaft by one piece composite one which resonate at higher speed and so keeping higher margin of safety. A drive shaft of composites offers excellent vibration damping, cabin comfort, reduction of wear on drive train components and increasing tyres traction. In addition, the use

of one piece torque tube reduces assembly time, inventory cost, maintenance, and part complexity. The first application of composite drive shaft to automotive was the one developed by Spicer U-joint divisions of Dana Corporation for the Ford econoline van models in 1985.

Polymer matrix composites such as carbon/epoxy or glass/epoxy offer better fatigue characteristics as micro cracks in the resin not growth further like metals but terminated at the holes of fibers. Generally composites have less susceptibility to the effect of stress concentration such as those caused by notches and holes, than metals.

Filament winding process is used in the fabrication of composite drive shafts. In this process, fiber tows wetted with liquid resin are wound over a rotating male cylindrical mandrel. In this technique the winding angle, fiber tension, and resin content can be varied. Filament winding is relatively inexpensive, repetitive and accurate in fiber placement.

An efficient design of composite drive shaft could be achieved by selecting the proper variables, which can be identified for safe structure against failure and to meet the performance requirements. As the length and outer radius of drive shafts in automotive applications are limited due to spacing, the design variables include the inside radius, layers thickness, number of layers, fiber orientation angle and layers stacking sequence. In optimal design of the drive shaft these variables are constrained by the lateral natural frequency, torsional vibration, torsional strength and torsional buckling. In this study another constraint is added in term of torsional fatigue to be employed in the design of drive shafts by the selection of the stacking sequence.

4.1 INTRODUCTION

Computer aided three dimensional interactive applications as high end CAD/CAE/CAM tool used worldwide.

CATIA V5 is developed by Dassault Systems. France is a completely re-engineered next generation family of CAD/CAM/CAE software solutions for product lifecycle management. Through its exceptionally easy to use state of the art user interface CATIA V5 delivers innovative technologies for maximum productivity and creativity from concept to the final product. CATIA V reduces the learning curve as it allows the flexibility of using feature based and parametric designs.

CATIA V5 provides three basic platforms – P1, P2 and P3. P1 is for small and medium sized process oriented companies which wish to grow towards the large scale digitized product definition. P2 is for the advanced design engineering companies that require product, process and resources modeling. P3 is for the high-end design application and is basically for automotive and aerospace industry where high equality surfacing or Class-A surfacing is used for designing.

The subject of interpretability offered by CATIA V5 includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit is that the links remain associative. As a result any changes made to this external data are notified and the model can be updated quickly.

CATIA V5 serves the basic tasks by providing different workbenches. A workbench is defined as a specific environment consisting of a set of tools which allows the user to perform specific design tasks in a particular area.

4.2 BASIC WORKBENCHES IN CATIA V5

- Part design workbench – the part design workbench is a parametric and feature based environment in which we can create solid models. The basic requirement of this is a sketch. The sketches for the objects are drawn in the sketcher workbench that can be invoked within the part design workbench by choosing the sketcher button from the sketcher toolbar. While drawing a sketch, various constrains are applied automatically.

We can also apply additional constrains and dimensions manually. After drawing the sketch exit the sketcher workbench and convert it into a feature. The tools in the part design workbench can be used to convert the sketch into a sketch based feature or we can apply the placed features such as fillets, chamfers… these features are called dress-up features

- Wireframe and surface design workbench – the wireframe and surface design workbench is also a parametric and feature based environment in which we can create wireframe or surface models. The tools in this workbench are similar to those in the part design workbench with the only difference that the tools in the environment are used to manipulate the surfaces to obtain the required shape.
- Assembly design workbench – the assembly design workbench is used to assemble the components using the assembly constrains available in this workbench. There are two types of assembly design approaches:
  1. Bottom up approach
  2. Top down approach

  In the bottom up approach of the assembly the previously created components are assembled together to maintain their design intent. In the top-down approach, components are created inside the assembly in the assembly design workbench.

- Drafting workbench – the drafting workbench is used for the documentation of the parts or assemblies in the form of drawing views and their detailing. The two type of drafting techniques are:
  1. Generative drafting
  2. Interactive drafting

  The generative drafting technique is used to generate the drawing views of parts and assemblies automatically. The parametric dimensions added to the component in the part design workbench during its creation can also be generated and displayed automatically in the drawing views. The generative drafting is bidirectional associative in nature. We can also generate the bill of materials and balloons to the drawing views.

  In interactive drafting we need to create the drawing views by sketching them using the normal sketching tools and then add the dimensions.

4.3 SALIENT FEATURES OF CATIA

- Feature based modeling – a feature is defined as the smallest building block that can be modified individually. A model created in CATIA V5 is a combination of number of individual features and each feature is related to the other directly or indirectly. These features understand their fit and function properly and therefore can be modified anytime during the design process. If proper design intent is maintained while creating the models then these features automatically adjust their values to any change in their surroundings. This provides greater flexibility to the design.
- Parametric modeling – the parametric modeling nature of a software package is defined as its ability to do the standard properties or parameters in defining the shape and size of geometry. The main function of this property is to derive the selected geometry to a new size or shape without considering dimensions. We can modify the shape and size of any feature at any stage of the design process. It makes the designing process very easy.
- Bidirectional associability – the bidirectional associability that exists between all the workbenches ensures that any modification made in the model in any case of the workbenches of CATIA V5, is automatically reflected in the other workbenches immediately.
- Easy accessible software
- Strong in 3d modeling
- Predefined shapes
- Powerful in surfacing
- User pattern facilities
- Supports both CSG and B-REP

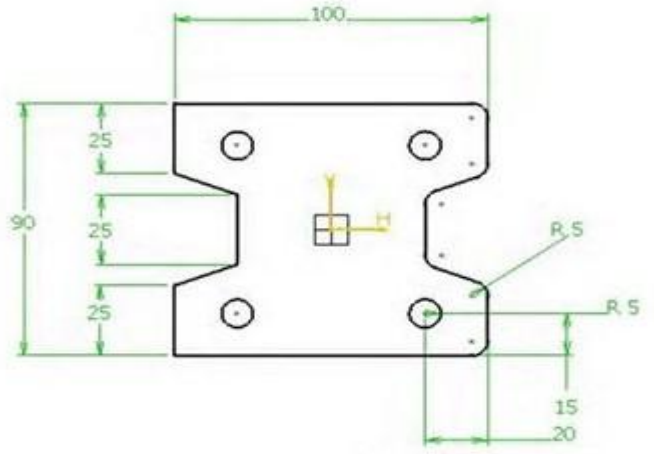
CHAPTER-5: MODELLING OF PROPELLER SHAFT ARRANGEMENT USING CATIA V5-R16

The propeller shaft arrangement of the Toyota quails vehicle consists of a propeller shaft, two centre blocks, a universal joint and a slip yoke. Modeling of each component IN CATIA V5 is as sequenced below.

5.1 MODELLING OF UNIVERSAL JOINT

The modeling steps of universal joint in CTAIA V5 are briefly discussed below.

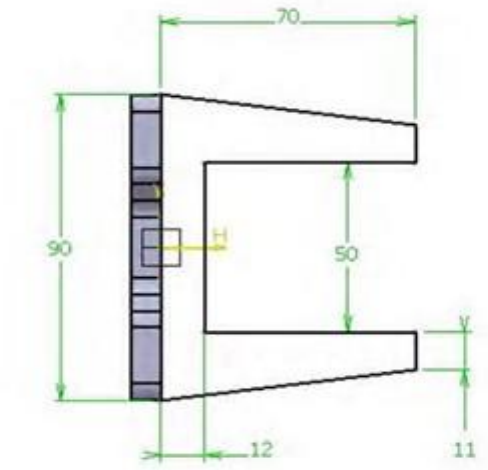
- First an arbitrary plane is selected and sketcher workbench in sketcher toolbar is invoked.
- Then the sketch shown in the figure 5.1 is drawn with appropriate dimensions and corners are filleted with radius of 5mm by invoking corner command in operation toolbar.



Front view Sketch of universal joint

Figure 5.1

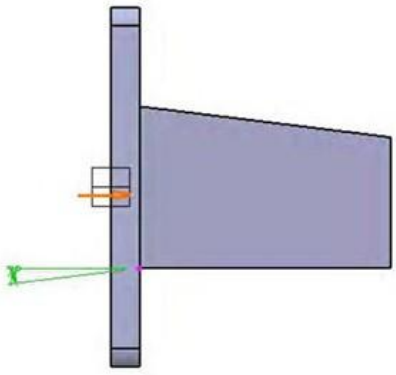
- Then the sketch is padded for 8mm by invoking pad command in sketch based features after exiting from the sketcher workbench.
- Next the view of the sketch is changed to side view and the sketch shown if the figure 5.2 is drawn by projecting the required edges (by invoking project 3D command in operations toolbar). And the sketch is padded by 22.5mm on both sides by invoking pad command in sketch based features toolbar.



Side view Sketch of universal joint

Figure 5.2

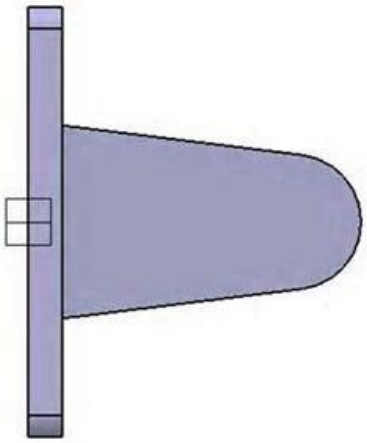
- Then the top and bottom surfaces are drafted by 5degrees each side by invoking draft command in dress-up features toolbar. This is shown in figure 5.3



Drafting top & bottom surfaces

Figure 5.3

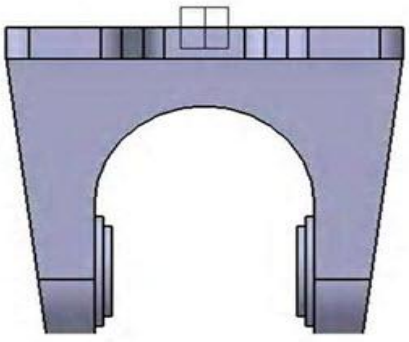
- Then the edge is chamfered by invoking tritangent fillet in dress-up toolbar. And the whole operations are mirrored to the other side by invoking mirror command in transformation features toolbar.



Filleting edges

Figure 5.4

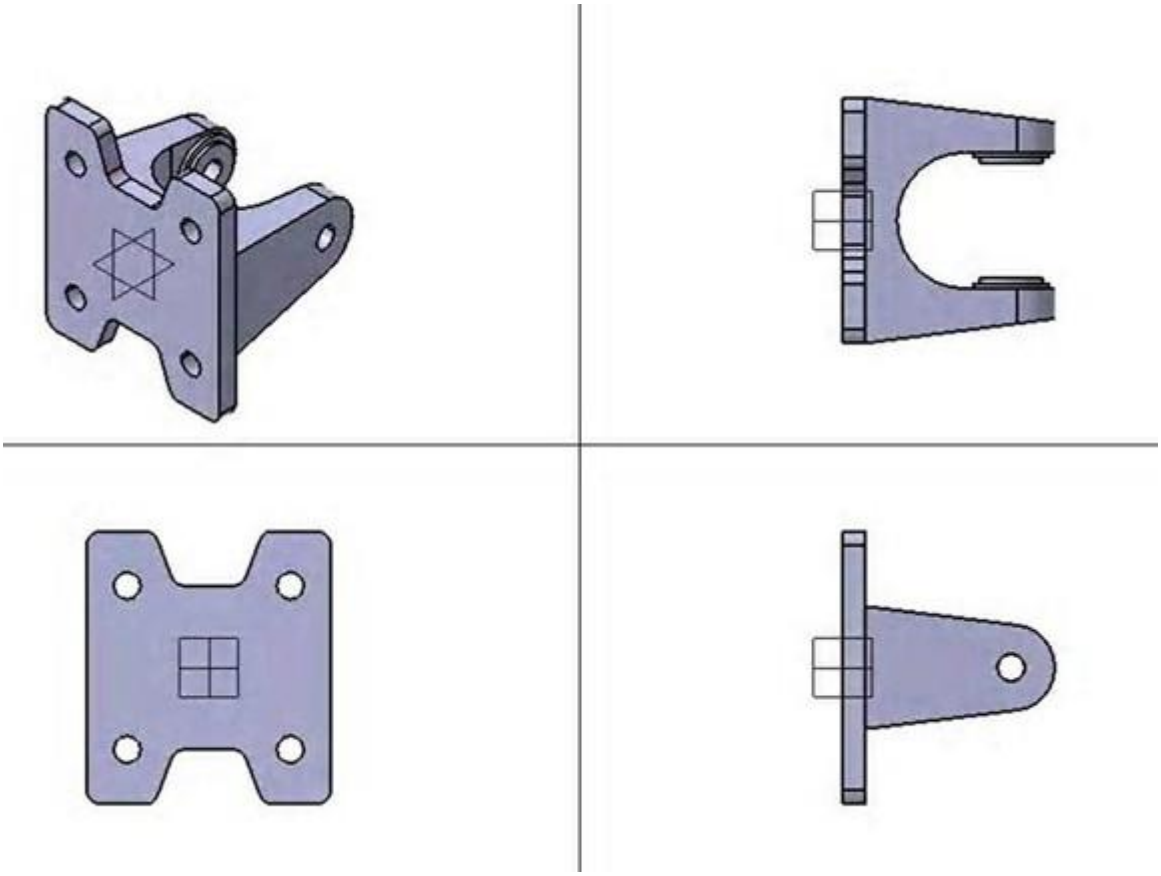
- Again tritangent fillet in dress-up toolbar is invoked to obtain the contour between the yokes as shown in the figure 5.5
- The circular projections of diameters 28mm and 23mm are obtained by invoking particular surface as working plane and doing pad operation of about 2mm each as shown in the figure 5.5



Creating circular projections

Figure 5.5

- A circle of 10mm diameter is pocketed on the yoke by drawing a circle of 10mm diameter on particular face and by invoking pocket command in sketch based features toolbar.
- Thus the universal joint yokes are designed and the complete figure is as shown in the figure 5.6.



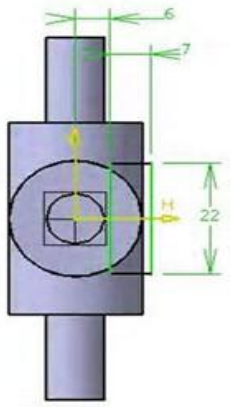
Various views of universal joint

Figure 5.6

5.2 MODELLING OF SPIDER (CENTER BLOCK)

The modeling steps of centre block in CTAIA V5 are briefly discussed below.

- First an arbitrary plane is selected and sketcher workbench in sketcher toolbar is invoked.
- Then a circle of 10mm diameter is drawn using drawing tools and sketcher is exited.
- Now the circle is padded about 36mm on both sides using pad command in sketch based features toolbar.
- Again the same plane is selected and entered into the sketcher to draw a circle of 23mm diameter. This circle is also padded about 19mm in both sides by invoking pad command in the sketch based features toolbar.
- Above three operations are repeated in any other plane which is perpendicular to the previously selected plane.
- Now the plane which is selected initially is selected and sketcher is invoked.
- A sketch as shown in the figure 5.7 is drawn. And this sketch is pocketed towards the centre block up to 12mm by invoking the pocket command in sketch based features toolbar.

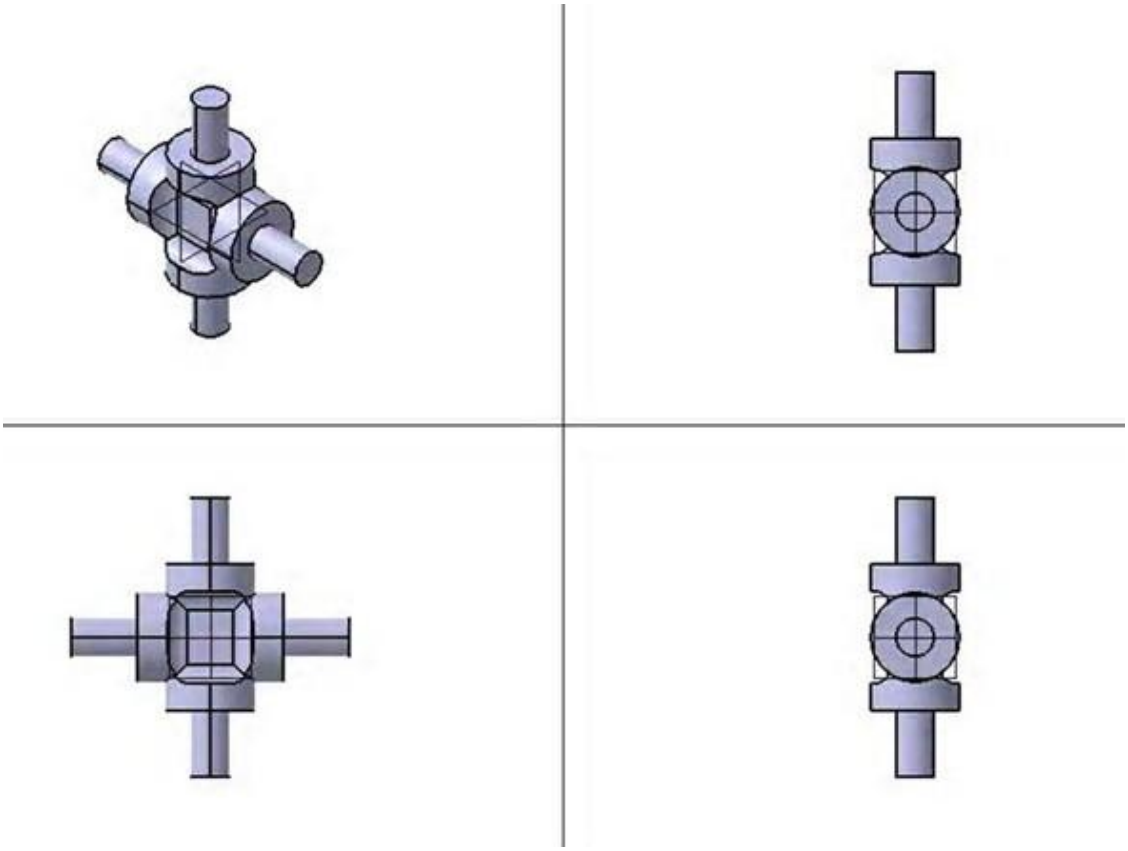


Side view of spider

Figure 5.7

- The edges are filleted about 5mm by invoking edge fillet command in dress-up features toolbar.

- The above two steps are mirrored by invoking mirror command in transformation features toolbar.
- Thus the centre block is modeled and is shown in the figure 5.8



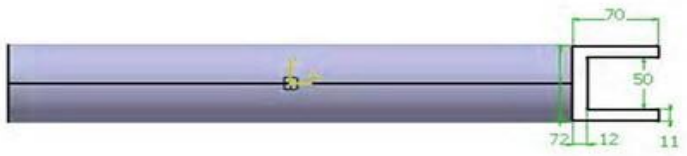
Various views of spider

Figure 5.8

5.3 MODELLING OF PROPELLER SHAFT

The modelling steps of propeller shaft in CTAIA V5 are briefly discussed below.

- First an arbitrary plane is selected and sketcher workbench in sketcher toolbar is invoked.
- A circle of diameter 75mm is drawn and is padded by invoking pad command in sketch based features toolbar.
- The side view of the shaft is made as the working plane and the sketch shown in the figure 5.9 is drawn by projecting required edges using project 3D command in operation toolbar.



Sketch for drive shaft flange flange

Figure 5.9

- This sketch is padded about 22.5mm on both sides using pad command in sketch based features toolbar.
- Now tritangent fillet is applied on both the yokes as shown in the figure 5.10.

Filleting edges

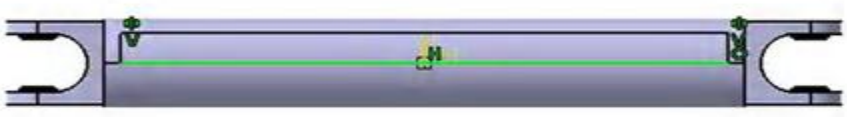
Figure 5.10

- The same tritangent fillet is made use to obtain the contour between the yokes. Ant the circular projections are obtained by invoking particular plane as working plane, drawing circles of diameters 28mm and 23mm and then finally padding about 2mm each by invoking pad command in sketch based features toolbar. This is shown in the figure 5.11

Creating circular projections

Figure 5.11

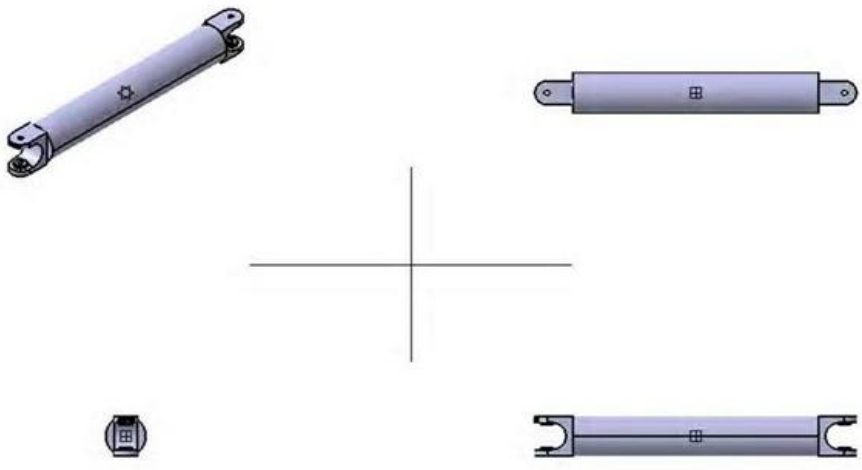
- A circle of 10mm diameter is pocketed on the yoke by drawing a circle of 10mm diameter on particular face and by invoking pocket command in sketch based features toolbar.
- The whole operations are carried out on the other side of the shaft.
- Finally a side of the shaft is selected as the working plane and the sketch shown in the figure 5.12 is drawn by making use of project 3d command in operation toolbar.



Front view of drive shaft

Figure 5.12

- This sketch is grooved for 180 degrees about the bottom edge by invoking the groove command in sketch based features toolbar. This forms an inner groove in the propeller shaft.
- This completes the modeling of the propeller shaft and is shown in the figure 5.13



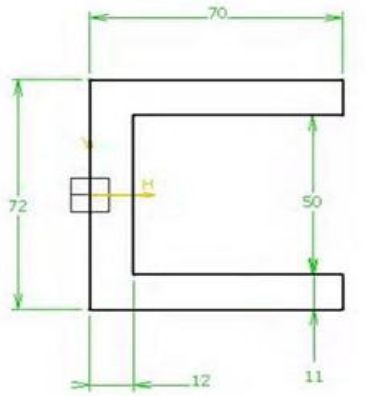
Various views of drive shaft

Figure 5.13

5.4 MODELLING OF SLIP YOKE

The modeling steps of slip yoke in CTAIA V5 are briefly discussed below.

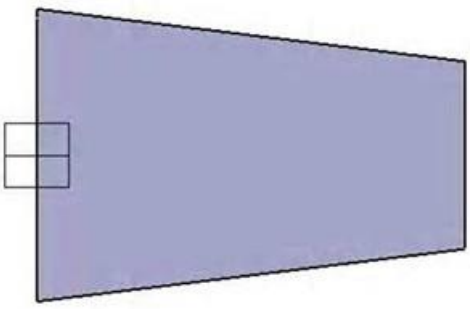
- First an arbitrary plane is selected and sketcher workbench in sketcher toolbar is invoked.
- The sketch shown in the figure 5.14 is drawn as per the dimensions and this sketch is padded about 22.5mm in both the sides by invoking pad command in the sketch based features toolbar.



Sketch for side view of slip yoke

Figure 5.14

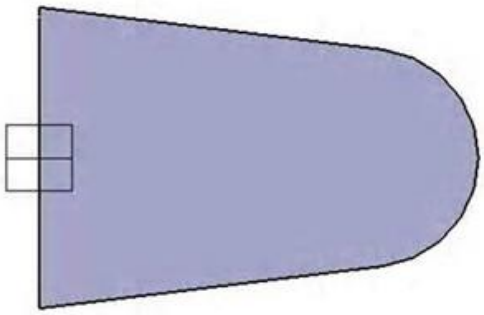
- The front and back faces of the yokes are drafted about 7 degrees towards the inner side by invoking the draft command in the dress-up features toolbar. This is shown in the figure 5.15.



Drafting surfaces

Figure 5.15

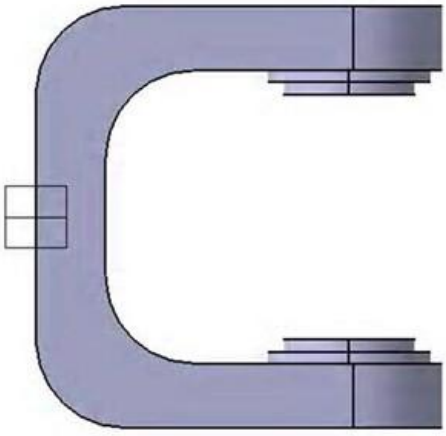
- Then the edges are filleted using tritangent fillet in dress-up features toolbar and this operation is mirrored to the other yoke by invoking mirror command in transformation toolbar. This is shown in figure 5.16



Filleting edges

Figure 5.16

- Now the sharp corners of the yoke edges are filleted about 15mm by invoking the edge fillet command in dress-up features toolbar.
- The circular projections of diameters 28mm and 23mm are obtained by invoking particular surface as working plane and doing pad operation of about 2mm each as shown in the figure 5.17

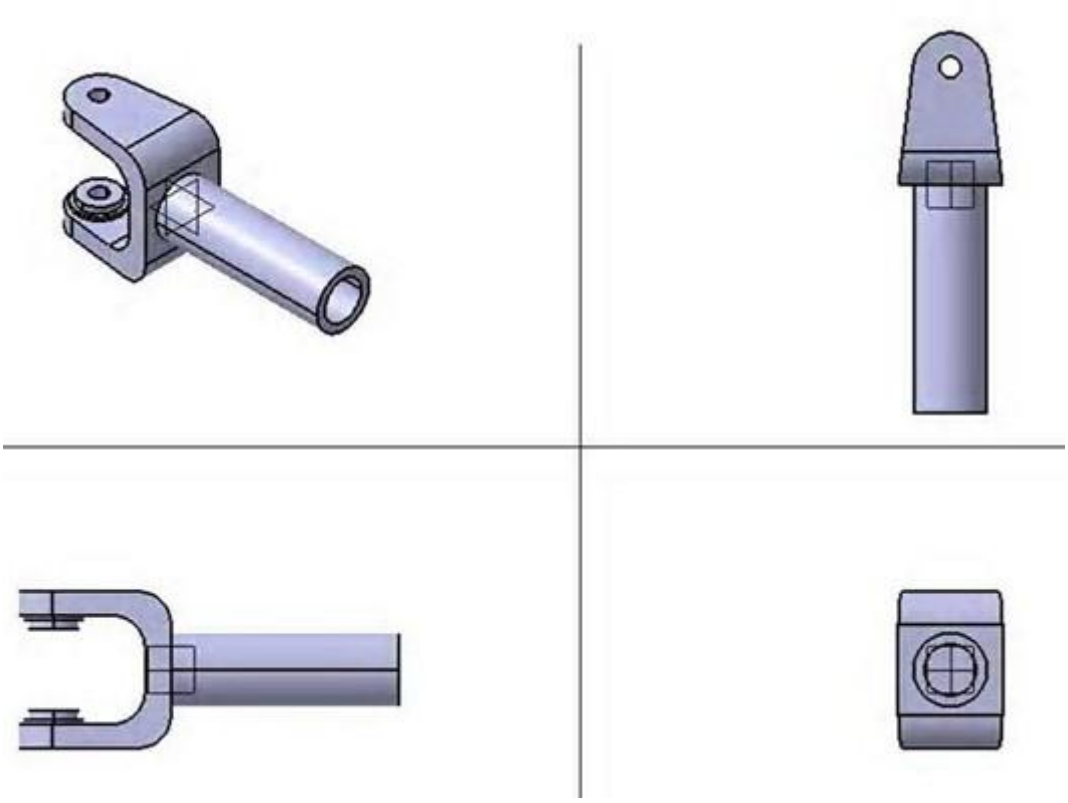


Creating circular projections

Figure 5.17

- A circle of 10mm diameter is pocketed on the yoke by drawing a circle of 10mm diameter on particular face and by invoking pocket command in sketch based features toolbar.

- The back of the slip yoke is selected as the working plane and circles of diameters 24mm and 34 mm are drawn to pad about 105mm by invoking pad command in the sketch based features toolbar.
- This completes the modeling of slip yoke and is shown in the figure 5.18



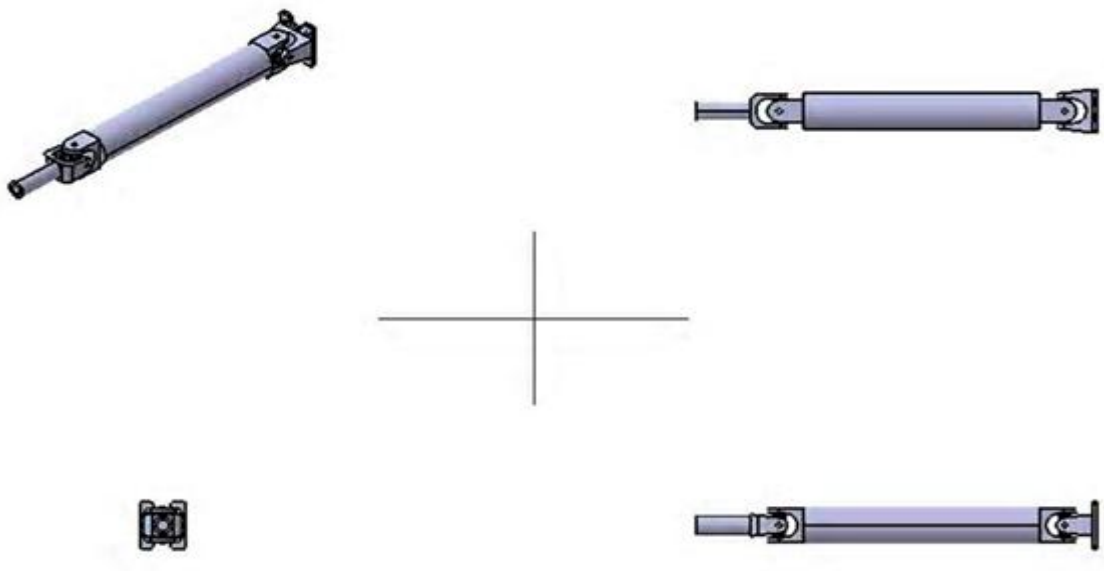
Various views of slip yoke

Figure 5.18

5.5 ASSEMBLY OF PROPELLER SHAFT ARRANGEMENT

The sequence how the propeller shaft arrangement is assembled is discussed below.

- CATIA V5 is opened and a new assembly file is created by navigation in to its start menu.
- Existing part command in product structure tools toolbar is invoked and one of the previously prepared part design (say propeller shaft) is added and its position is fixed using constrains position toolbar.
- Similarly all other components are added one by one and assembled using the coincidence, offset and parallelism constrains in constrains position toolbar.
- This completes the assembly of propeller shaft arrangement of Toyota qualis and is shown in the figure 5.19



Various views of assembly of propeller shaft

Figure 5.19

CHAPTER-6: FINITE ELEMENT METHOD

6.1 INTRODUCTION

The finite element method is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industries. In more and more engineering situations today, we find it is necessary to obtain approximate solutions to problems rather than exact closed from solution.

It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solution is a mathematical expression that gives the value of the desired unknown quantity of at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineering study shifts towards numerical methods that provide approximate, but acceptable solutions.

The finite element method has become powerful tool for the numerical solution of a wide range of engineering problems. It has developed simultaneously with the increasing use of the high-speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis.

The fundamental areas that have to be learned for working capability of finite element method include:

- **Matrix algebra.**
- **Solid mechanics.**
- **Variational methods.**
- **Computer skills.**

Matrix techniques are definitely most efficient and systematic way to handle algebra of finite element method. Basically matrix algebra provides a scheme by which a large number of equations can be stored and manipulated. Since vast majority of literature on the finite element method treats problem in structural and continuum mechanics, including soil and rock mechanics, the knowledge of these fields became necessary. It is useful to consider the finite element procedure basically as variational approach.

The term “finite element “distinguishes the technique from the use of infinitesimal “Different elements” used in calculus, difference equations, for which although the steps into which space is divided into finite elements are finite in size, there is a little freedom in the shapes that the discrete steps can take. FEA is a way to deal with structures that are more complex than dealt with analytically using the partial differential equations and gives answers to the “real world” structural problems. It has been substantially extended scope during the roughly forty years of its use.

FEA makes is possible to evaluate a detail and complex structure, in a computer during the planning of the structure. The demonstration in the computer about the adequate strength of the structure and possibility of improving design during planning can justify the cost of this analysis work. FEA has also been known to increase the rating of the structures that were significally over designed and build many decades ago.

In the absence of finite element analysis (or other numerical analysis), development of structures must be based on hand calculations only. For complex structures, the simplifying assumptions are required to make any calculations possible can lead to a conservative design and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. Significant changes in design involve expensive strain gauging to evaluate strength and deformation.

6.2 GENERAL PROCEDURE OF FEA

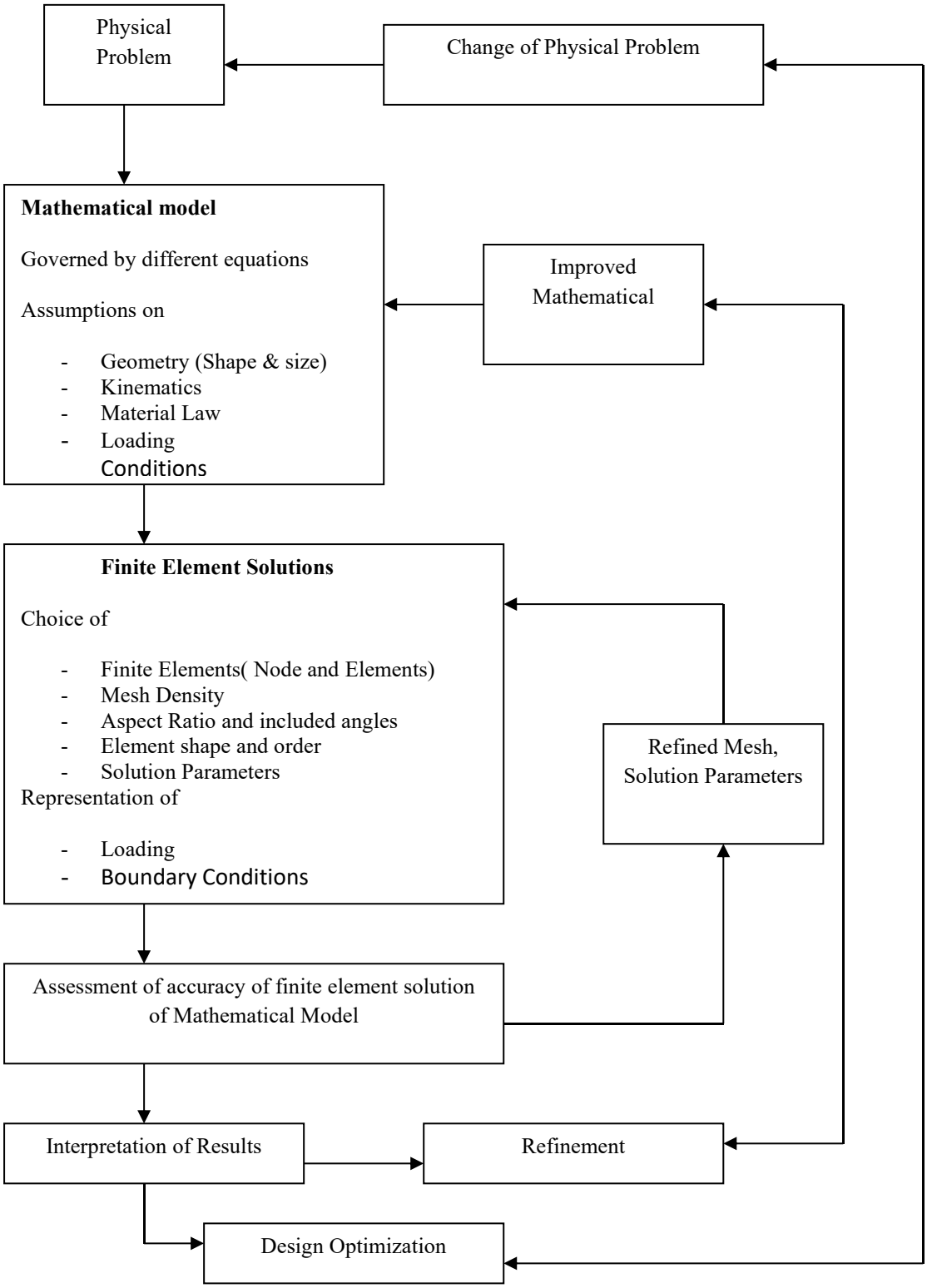
The following steps summarize the general procedure for finite element analysis.

- STEP 1 - The continuum is a physical body, structure or solid being analyzed. Discretization may be simply described as process by which the given body is subdivided into equivalent system of finite elements...
- STEP 2 - The selection of displacement or temperature models or shape functions representing approximately the actual distribution of the displacement or temperature. The three factors which influence the selection of shape functions are

a. The type and degree of displacement model
b. Displacement magnitudes
c. The requirements to be satisfied which ensuring correct solution.

- STEP 3 - The derivation of the stiffness matrix which consists of the coefficients of the equilibrium equations derived from the geometric and material properties of the element. The stiffness relates the displacement at nodal points to applied forces at nodal points.
- STEP 4 - Assembly of the algebraic equations for the overall discredited continuum includes the assembly of overall stiffness matrix for the entire body from individual element stiffness matrices and the overall global load vector from the elemental load vectors.
- STEP 5 - The algebraic equations assembled in step 4 are solved for unknown displacements by imposing the boundary conditions. In linear equilibrium problems, this is a relatively straightforward application of matrix algebra techniques.
- STEP 6 - In this step, the element strains and stresses are computed from the nodal displacements that are already calculated from step 5.

6.3 THE PROCESS OF F.E.A:



6.4 ADVANTAGES OF FEM:

The FEM is based on the concept of discretizitation. Nevertheless as either a variational or residual approach, the technique recognizes the multi dimensional continuity of the body not only does the idealization portray the body as continuous but it also requires no separate interpolation process to extend the approximate solution to every point within the continuum. Despite the fact that the solution is obtained at a finite number of discrete node points, the formation of field variable models inherently provides a solution at all other locations in the body. In contrast to other variational and residual approaches, the FEM does not require trial solutions, which must all apply to the entire multi dimensional continuum. The use of separate sub-regions of the finite elements for the separate trial solutions thus permits a greater flexibility in considering continua of the shape.

Some of the most important advantages of FEM derive the techniques of introducing boundary conditions. This is another area in which they differ from other variational or residual approaches. Rather than requiring every trial solution to satisfy the boundary conditions, one prescribes the conditions after obtaining the algebraic equations for assemblage. No special techniques or artificial devices are necessary, such as the noncentered difference equations of factious external points often employed in the finite difference method.

The FEM not only accomplishes complex geometry and boundary conditions, but it also has proved successful in representing various types of complicated material properties that are difficult to incorporate into other numerical methods. For example, Formulations in solid mechanics have been devised for anisotropic, non linear, hysteretic, time dependant or temperature dependant material behavior.

Finally, an engineer may develop a concept of the FEM at different levels. It is possible to interpret the method in physical terms. On the other hand, the method may be explained entirely in mathematical terms. The physical or intuitive nature of the procedure is particularly useful to the engineering student and practicing engineer.

6.5 LIMITATIONS OF FEM:

One limitation of the finite element method is that a few complex phenomena are not accomplished adequately by the method at its current state of development. Some examples of such phenomena from the realm of solid mechanics are cracking and fracture behavior, contact problems, bond failures of composite materials, and non-linear material behavior with soft working. Another example is transient, unconfined seepage problems. The numerical solution of propagation or transient problem is not satisfactory in all respects. Many of these phenomena are presently under research, and refinements of the method to accommodate these problems better can be expected.

The finite element method has reached a high level of development as a solution technique however, the method yield realistic results only if coefficients or material parameters which describe the basic phenomena are available. Material non-linearity in solid mechanics is a notable example of a field in which our understanding of the material behavior has lagged behind the development of the analytical tool. In order to exploit fully the power of the finite element method, significant effort must be directed toward the development of suitable constitutive laws and the evaluation of realistic coefficients and material parameters.

Even the most efficient finite element computer codes require a relatively large amount of computer memory and time. Hence, use of the method is limited to those who have access to relatively large, high speed computers. The most tedious aspect of the use of finite element method is the basic processes of subdividing the continuum and of generating error-free input data for the computer. Although these processes may be automated to a degree, they have been totally accomplished by computer because some engineer judgment must be employed in the discretizitation. Error in the input data may go undetected and the erroneous result obtained there from may appear acceptable. Consequently, it is essential that the engineer/programmer provide checks to detect such errors.

Finally, as for any approximate numerical method, the results of a finite element analysis must be interpreted with cadre. We must be aware of the assumptions employed in the formulation, the possibility of numerical difficulties, and the limitations in the material characterizations used. A large volume of solution information is generated by finite element routine, but this data is worthwhile only when its generation and interpolation are tempered by proper engineering judgment.

6.6 APPLICATIONS OF FINITE ELEMENT METHODS:

There are wide areas of application of finite element method are listed below:

S.NO	FIELDS	TYPICAL EXAMPLES
1.	Structural mechanics(deflection & stress analysis of structures) **A.** 2- D analysis **B.** 3- D analysis **C.** Bending of plates	**A.** In plane stress, stretching of plates, gravity dams, axisymmetric solid shells, rocket, motors machine parts such as shafts, beams, bridges etc., **B.** 3-D trusses, spaces frames such as cranes, thin walled structure like machine tools, transmission towers, nuclear towers, nuclear reactors ship structures , radar domes, building dams, shell roofs, arches drilling platforms etc., **C.** Floor slabs, thin walls of machine tool structure, ship decks, aircraft and spacecraft panels.
2.	Soil and rock mechanics	Foundation layers, rock joints, pavements stability of excavation such as river blanks, embankments, open pit and underground mining problems etc.,
3.	Thermal analysis and fluid mechanics	Transient and steady state temperature distribution, thermal strain and stresses in mechanical and civil structures. Flow towards wells, seepage through foundations. Fluid flows in pipes, canals. Etc.,
4.	Hydro-elasticity	Hydrodynamic and air bearings, reservoir-dam interactions, sloshing of liquids in flexible containers.
5.	Dynamics	Natural frequencies and mode shapes of structures. Responses to arbitrary dynamic lading such as wind explosions, water waves, earth quakes. Etc….

6.7 BASIC APPROACH TO FEA:

Basic approach for any finite element analysis (FEA) can be divided in to three parts

6.7.1 Pre – processor

6.7.2 Solver

6.7.3 Post – processor

6.7.1 PRE– PROCESSOR:

Pre – processor mainly contains building of model, meshing, assigning material properties etc.

Building a model:

Geometry is usually difficult to describe, as it has to be as close as real. Since it has to take real world loads and boundary conditions, it is equal to proto type simulated in a computer.

Creation of finite element model of meshing:

After meshing material properties and structural properties to the model, meshing is done. Meshing means dividing a solid model into a finite number of finite sized elements.

FEA uses a complex system of points called nodes, which make a grid called MESH. This mesh is programmed to contain material and structural properties, which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions that will receive large amounts of stress usually have a higher node density than those, which experience little or no stress.

Points of interest may consist of fracture of previously tested material, fillets, corners, complex detail and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material to the object, creating many elements. Each FEA programs may come with an element library or one is constructed over time. Elements are thought to fill the whole experiment hence choice of element is very critical.

While meshing it has to be decided whether we have to go for finer mesh or coarser one. The finer the mesh, the better will be the result but longer analysis time. Therefore, a compromise between accuracy and solution speed is usually made. The mesh may be created manually or automatically. In the manually created mesh, we will notice that the elements are smaller at joints. This is known as mesh refinement and it enables the stress to be captured at t he geometric distributions. (The junction).

Manual meshing is a tedious process for models with any degree of complications, but inbuilt mesh engine creates mesh automatically thus reducing the time. These automatic meshing has limitations as to regards mesh quality and solution accuracy.

6.7.2 SOLVER :

Solvers are geometric task oriented. These are developed for specific applications. Solvers are designed based on continuum approach where in construction of mass, momentum and energy equations of state, thermodynamic equations as when required for each of the elements and the solution is obtained by interpreting these solutions. The solution to these equations depends on two methods

1. Implicit.
2. Explicit

Choice of the method is based on the nature of the problems.

The main goal of a finite element analysis is to execute a structure or a component response to certain loading conditions. Therefore specifying the proper loading conditions is a key in the analysis. Theses loading conditions may be static, dynamic, transient whose nature may be linear or non-linear.

The word loads in FEA includes the applied forcing functions externally or internally. For example loads in structural are displacement, forces, pressures, temperatures and gravity. In thermal analysis the applied loads are temperature; heat flow rates convections and heat generated.

Solver reads the model created by the preprocessor and formulates the mathematical representation of the model. All the parameters defined in the preprocessing stage are used to do this. If the model is correct the solver proceeds to form the element stiffness matrix for the problem. After calculating the element stiffness matrix it calculates the results like

displacements, pressure, stresses, strains etc., at node within the component or continuum. All these results are sent to a result file, which may be read by the post-processor.

6.7.3 POST-PROCESSOR:

Here the results of the analysis are read and interpreted. They can be presented in the form of table, a contour plot, deformed shape of the component or the mode shapes and natural frequencies are involved. Other results are available for fluids, thermal and electrical analysis types. Contour plots are usually the most effective way of viewing results for structural type problems. Slices can be made through 3 - D models to facilitate the viewing of internal stress patterns.

Post processor includes the calculations of stress and strains in any one of the x, y, and z – directions or induced in a direction at an angle to the co-ordinate axis. The principle stresses and strains may also be plotted or if required the yield stress and strains according to the main theories of failure (Von-mises, st-venants, trescas etc) can be plotted. Other information such as the strain energy, plastic strain and creep strain may be obtained for certain type of analysis.

6.8 POPULAR FEA SOFTWARES

There are varieties of commercial FEA software available over the market. No single software is supposed to have all the capabilities that can meet the complete simulation requirements of a design. Hence based upon the requirements, some of the firms use one or more FEA software. While some other firms develop their own customized versions of software. Some of the popular commercially available FEA software's are as follows.

- Adina
- Abaqus
- Ansys
- MSC/Nastran
- Cosmos
- NISA
- Marc
- Ls-Dyna
- MSC/Dytran
- Star-CD

7.1 INTRODUCTION

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems. It enables engineers to perform the following tasks - build computer models or transfer cad models of structures, products, components or system, apply operating loads or other design performance conditions, study physical responses such as stress levels, temperature distributions or electromagnetic fields, optimize a design early in the development process to reduce production costs, carryout prototype testing in environment where it otherwise would be undesirable or impossible.

7.2 SPECIFIC CAPABILITIES OF ANSYS

- **Structural Analysis**
- **Static Analysis**
- **Transient Dynamic Analysis**
- **Buckling Analysis**
- **Thermal Analysis**
- **Fluid Flow Analysis**
- **Magnetic field analysis**
- **Acoustics / Vibration**
- **Coupled Fields**

7.3 STRUCTURE OF ANSYS

In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

- Preprocessing: This stage deals with defining the problem. The major steps in preprocessing are given below:
- Define key points/lines/areas/volumes
- Define element type and material/geometric properties
- Mesh lines/areas/volumes as required.

The amount of details required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axi-symmetric, 3D).

- Solution: assigning loads, constraints and solving; Here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting set of equations.
- Post processing: further processing and viewing of the results; In this stage one may wish to see:
- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagrams

7.4 STEP BY STEP PROCESSING OF GOOD ANALYSIS

It is important to think about the entire process up front because it's very easy to get wound up in the details of doing an analysis and lose sight of the big picture. The list below outlines the steps that are to be followed.

- Thoroughly understand the actual problem
- Predict what you think the answer will be
- Decide if finite element analysis is a reasonable method for analyzing this problem
- Determine the type of analysis needed to obtain reasonable answers
- Determine the type of elements you will use
- Determine the geometry needed to generate the elements
- Create the geometry within Ansys or import it from another source
- Create the attributes needed to define the elements
- Set element sizes
- Mesh the geometry and create any other elements that are needed
- Apply boundary conditions
- Set the load step controls
- Write the load step files
- Solve the load step files
- Review the results

- Interpret the results
- Compare the results to your original prediction
- Iterate as needed to obtain a satisfactorily accurate answer

7.5 ADVANTAGES OF ANSYS

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product life before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

CHAPTER-8: ANALYSIS ON PROPELLER SHAFT ASSEMBLY OF TOYOTA QUALIS

8.1 ANSYS WORKBENCH

To carry out the analysis on the propeller shaft assembly, ANSYS WORKBENCH mode is used which is one of the auxiliary modes provided along with ANSYS 11.0 version. The key features which make ANSYS WORKBENCH dominated over conventional classical mode of ANSYS are

- No need to define element type – like in conventional classic mode, there is no need of remembering bulk data regarding the element type to be defined for an analysis of a model.
- Less mesh time – it is one of the most important key features of the utility. ANSYS workbench provides ease of use by taking very less time for meshing even for a large mesh density.
- Importing complete details of modeling – conventional classic mode of ANSYS has a drawback of missing some of the modeling identities while exporting modeling files. This can be overcome in this mode and exports a complete geometry and all the modeling features.
- Less analysis time – problems having very high mesh density can be solved in ANSYS WORKBENCH within very less time compared to conventional classic mode. This feature plays a major role in problems involving large structures and/or high mesh density which consumes a lot of time when solving in conventional classic mode.
- Ease of use – it is the most effective feature of workbench mode. The tree mode display of analysis procedure makes the self justification over the current problem and displays tips and guides the analysis procedure to lead to a better solution. This feature makes the workbench mode very simplified in use compared to the conventional classic mode.
- Along with these features it had another feature. That is modeling. This feature acts as powerful tool in carrying out modeling as that of in popular modeling packages viz.. CATIA, PRO/E… this eliminates the use of another modeling tool and the whole work starting from scratch can be carried out in this mode.

8.2 ANALYSIS PROCEDURE

- Save the modeled file that is prepared in CATIA or PRO/E packages to appropriate format that is supported by ANSYS. ANSYS supports sat, agdb, model, dlv, CATIA Part, CATIA Product, tin, ipt, iam, igs, iges… the models created in doing this project are saved in stp format.
- Open the ANSYS WORKBENCH mode and select simulation mode. This takes to a simulation mode where model files can be imported and different analysis can be done on the problem.
- First the material properties are to be defined. For this change the current tab to project and select the current file and tick the material properties and click on the engineering data icon located just below the standard toolbar. This opens a new tab named engineering data. Enter the required properties of the material. Here lies a material library in which some standard materials are saved with their properties. These parameters can be exported or can be directly entered.
- Now change the tab to simulation tab. Select geometry icon in the toolbar and export the model file saved in appropriate format. This imports the modeling file into the simulation mode. Now check whether the geometry is ticked or not to ensure that all the modeling properties are imported or not.
- By importing geometry, the new branch named mesh is automatically displayed in the tree located left. By right clicking on mesh, size of mesh can be defined. In this analysis the size of mesh is defined as 0.01mt. To generate mesh right click on mesh and click on generate mesh.
- After meshing, select new analysis and select structural analysis. This generates a branch with name structural analysis in the tree. But this branch is tagged with a question mark which indicates that the required parameters to carry out the structural analysis are not yet defined. The two basic constrains that are to be defined for structural analysis are fixed supports and force. To define these constrains, right click the structural analysis branch and insert fixed support and moment. This creates two sub-branches named fixed support and moment. To define these properties select those properties and select the faces of the model on which the particular property is to be applied. After selecting the appropriate faces, select apply in the left bottom table that under geometry. And give values if any after defining the load type

(whether vector type or component type). Here in this analysis, fixed supports are given at four holes located on the yokes of universal joint and a rated torque of 15.4 NM (maximum torque of Toyota quails is 15.4 KgM @ 2400 rpm) is applied about X-axis inside the slip yoke. After defining these two properties a tick mark can be observed at the branch structural analysis indicating to proceed to solution phase.

- The next phase is the solution phase. The different results that are required are to be inserted in the solution branch. For this right click on the solution branch and insert required parameters to be analyzed. After doing this, by clicking on the solve icon located on the toolbar, solution can be obtained. To view the results click on the required parameter.
- Workbench has a feature to capture images, record video and point maximum and minimum values with a few clicks. Thus required data can be stored in required format for further reference.
- Workbench also has another feature. This generates an automatic report of the current analysis. This can be obtained just by clicking on the report preview tab located just below the image of the current object. This can also be exported to word or excel file.
- The same procedure is followed to carry out the analysis on different composite materials merely changing the properties of the materials in each analysis.

The same analysis is carried out on the propeller shaft arrangement by changing the materials each time. The materials and their results are discussed in next sessions.

8.3 MATERIALS USED IN ANALYSIS AND THEIR PROPERTIES

The materials that were used in this analysis are from epoxy family. Epoxy is the most common type of matrix material due to its

- High strength
- Low viscosity and low flow rates, which allow good wetting of fibers and prevent misalignment of fibers during processing.
- Low volatility during cure.
- Low shrink rates
- Available in more than 20 grades to meet specific property and processing requirement.

The properties of those materials are

Structural Steel	
Young's Modulus	2.07e+011 Pa
Poisson's Ratio	0.3
Density	7600. kg/m³
Allowable stress	370e+006 pa
E Glass	
Young's Modulus X direction	5.e+010 Pa
Young's Modulus Y direction	1.2e+010 Pa
Young's Modulus Z direction	1.2e+010 Pa
Major Poisson's Ratio XY	0.3
Major Poisson's Ratio YZ	0.3
Major Poisson's Ratio XZ	0.3
Shear Modulus XY	5.6e+009 Pa
Shear Modulus YZ	5.6e+009 Pa
Shear Modulus XZ	5.6e+009 Pa
Density	2000. kg/m³
Allowable stress	400e+006 Pa

E Carbon	
Young's Modulus X direction	1.9e+011 Pa
Young's Modulus Y direction	7.7e+009 Pa
Young's Modulus Z direction	7.7e+009 Pa
Major Poisson's Ratio XY	0.3
Major Poisson's Ratio YZ	0.3
Major Poisson's Ratio XZ	0.3
Shear Modulus XY	4.2e+009 Pa
Shear Modulus YZ	4.2e+009 Pa
Shear Modulus XZ	4.2e+009 Pa
Density	1600. kg/m³
Allowable Stress	440e+006 Pa
E Glass Polyester Resin	
Young's Modulus X direction	3.4e+010 Pa
Young's Modulus Y direction	6.53e+009 Pa
Young's Modulus Z direction	6.53e+009 Pa
Major Poisson's Ratio XY	0.217
Major Poisson's Ratio YZ	0.366
Major Poisson's Ratio XZ	0.217
Shear Modulus XY	2.433e+009 Pa
Shear Modulus YZ	1.698e+009 Pa
Shear Modulus XZ	2.433e+009 Pa
Density	2100. kg/m³
Allowable Stress	420e+006 Pa

Since the ANSYS WORKBENCH has a special feature to generate automatic reports of the carried out analysis, those reports are included in this section

8.4 WEIGHT CALCULATIONS:

Volume of Universal joint = 0.0001726m³

Volume of Spiders = 0.00005166 m³

Volume of Shaft = 0.002 m³

Volume of slip yoke = 0.0001504 m³

Volume of total Assembly = 0.00255 m³

Weight of assembly = Volume×Density×9.81(N)

FOR STEEL:

Weight of assembly = 0.00255×7600×9.81

= 177.13N

FOR E-GLASS:

Weight of assembly = 0.00255×2000×9.81

= 46.62N

FOR E-CARBON

Weight of assembly = 0.00255×1600×9.81

= 37.3N

FOR E-GLASS POLYESTER

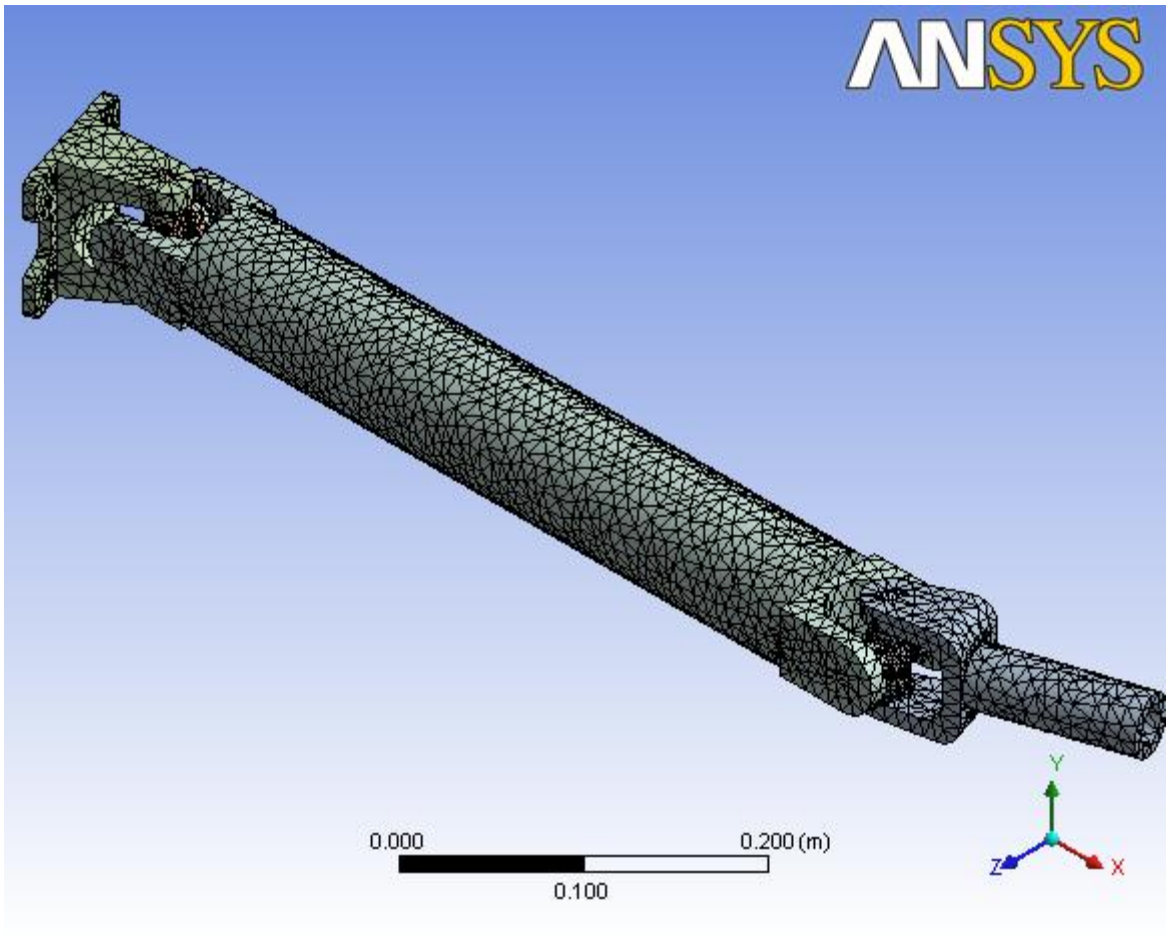
Weight of assembly = 0.00255×2100×9.81

= 48.932N

8.5 MESHING OF ASSEMBLY

Meshing Element = Triangle

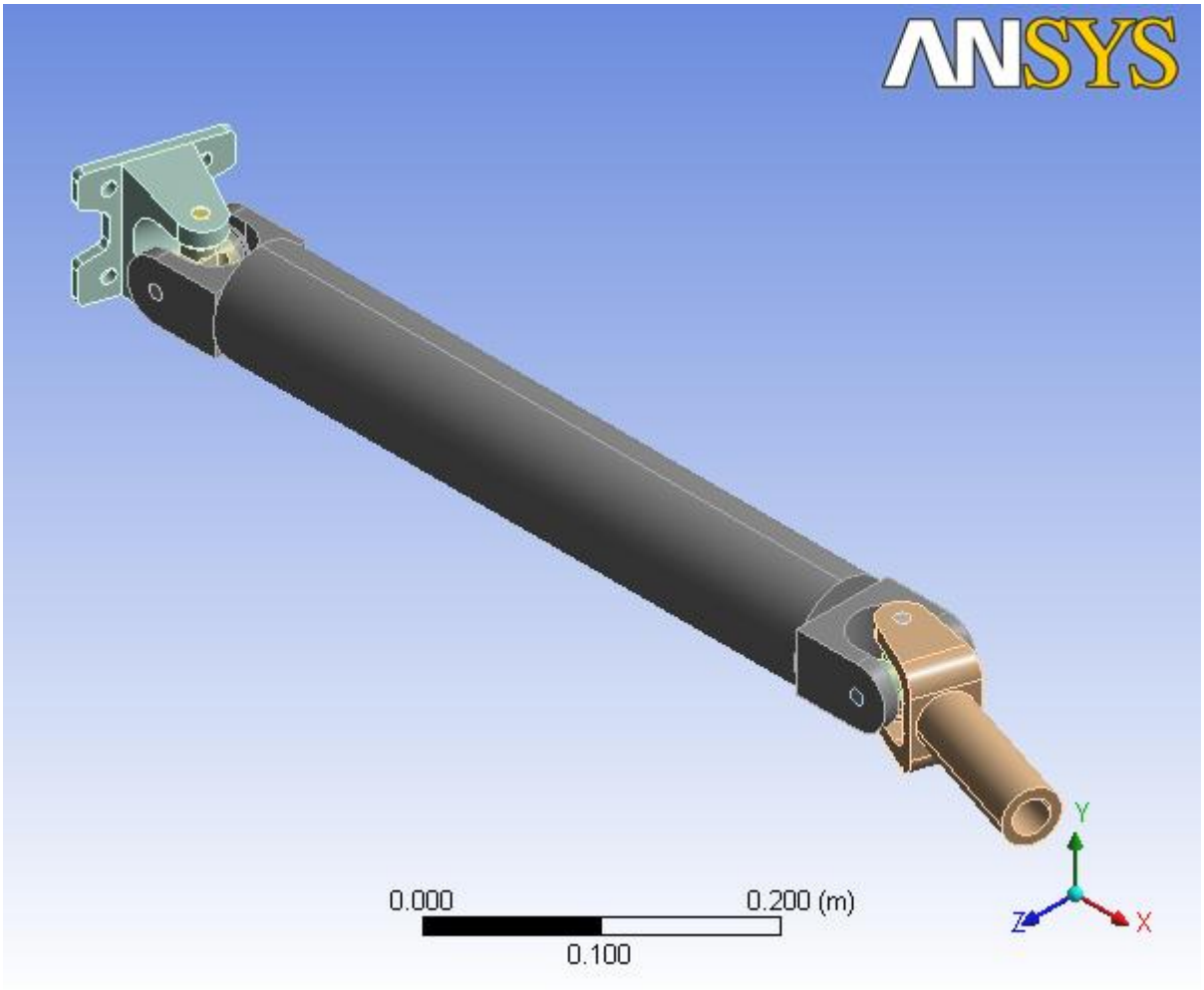
Element length = 0.02m



8.6 ANSYS GENERATED REPORT ON STRUCTURAL STEEL

Project

First Saved	Tuesday, March 23, 2010
Last Saved	Tuesday, March 23, 2010
Product Version	11.0 Release



Contents

- **Model**
  - Geometry
    - Parts
  - Connections
    - Contact Regions
  - Mesh
    - Body Sizing
  - **Static Structural**
    - Analysis Settings
    - Loads
    - Solution
      - Solution Information
      - Results
- **Material Data**
  - Structural Steel

Units

TABLE 1

Unit System	Metric (m, kg, N, °C, s, V, A)
Angle	Degrees
Rotational Velocity	rad/s

Model

Geometry

TABLE 2
Model > Geometry

Object Name	Drive shaft assembly
State	Fully Defined
Definition	
Source	E:\rafi\universal coupling with angles.igs
Type	Iges
Length Unit	Meters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Length X	0.78804 m
Length Y	9.e-002 m
Length Z	0.13716 m
Properties	
Volume	2.6274e-003 m³
Mass	20.625 kg
Statistics	
Bodies	9
Active Bodies	9
Nodes	15973
Elements	7650
Preferences	
Import Solid Bodies	Yes

Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	Yes
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	No
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

TABLE 3
Model > Geometry > Parts

Object Name	*Part 1*	*Part 2*	*Part 3*	*Part 4*	*Part 5*
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Material	Structural Steel				
Stiffness Behavior	Flexible				

Nonlinear Material Effects	Yes				
Bounding Box					
Length X	1.e-002 m		2.3e-002 m	0.18034 m	0.10491 m
Length Y	7.2e-002 m	1.e-002 m	3.8e-002 m	7.2e-002 m	9.e-002 m
Length Z	1.e-002 m	7.2e-002 m	3.8e-002 m	9.6876e-002 m	0.11869 m
Properties					
Volume	5.6546e-006 m³	5.6543e-006 m³	1.3979e-005 m³	1.5485e-004 m³	1.7635e-004 m³
Mass	4.4388e-002 kg	4.4386e-002 kg	0.10973 kg	1.2156 kg	1.3844 kg
Centroid X	0.5075 m		0.50745 m	0.56947 m	-8.6135e-002 m
Centroid Y	-5.183e-017 m	-3.9265e-019 m	8.8326e-006 m	1.2729e-007 m	-2.1067e-003 m
Centroid Z	-1.3363e-017 m	-3.145e-017 m	4.1988e-006 m	2.0135e-002 m	-1.2553e-002 m
Moment of Inertia Ip1	1.9341e-005 kg·m²		2.5141e-005 kg·m²	6.4976e-004 kg·m²	1.5551e-003 kg·m²
Moment of Inertia Ip2	5.4763e-007 kg·m²	1.9342e-005 kg·m²	1.6331e-005 kg·m²	2.7237e-003 kg·m²	9.6254e-004 kg·m²
Moment of Inertia Ip3	1.9342e-005 kg·m²	5.4763e-007 kg·m²	1.6034e-005 kg·m²	3.0689e-003 kg·m²	1.6583e-003 kg·m²
Statistics					
Nodes	122		1806	2331	2880
Elements	12		857	1105	1323

TABLE 4
Model > Geometry > Parts

Object Name	*Part 6*	*Part 7*	*Part 8*	*Part 9*
State	Meshed			
Graphics Properties				
Visible	Yes			
Transparency	1			
Definition				
Suppressed	No			
Material	Structural Steel			
Stiffness Behavior	Flexible			
Nonlinear Material Effects	Yes			
Bounding Box				
Length X	1.e-002 m		2.3e-002 m	0.6 m
Length Y	7.2e-002 m	1.e-002 m	3.8e-002 m	7.5e-002 m
Length Z	1.e-002 m	7.2e-002 m	3.8e-002 m	7.5e-002 m
Properties				
Volume	5.6546e-006 m³	5.6543e-006 m³	1.3979e-005 m³	2.2456e-003 m³
Mass	4.4388e-002 kg	4.4386e-002 kg	0.10973 kg	17.628 kg
Centroid X	-4.75e-002 m		-4.7549e-002 m	0.22988 m
Centroid Y	-5.183e-017 m	-3.9265e-019 m	8.8326e-006 m	9.0071e-009 m
Centroid Z	-6.2579e-018 m	-2.4344e-017 m	4.1988e-006 m	-4.3077e-008 m
Moment of Inertia Ip1	1.9341e-005 kg·m²		2.5141e-005 kg·m²	1.2443e-002 kg·m²
Moment of Inertia Ip2	5.4763e-007 kg·m²	1.9342e-005 kg·m²	1.6331e-005 kg·m²	0.39546 kg·m²
Moment of Inertia Ip3	1.9342e-005 kg·m²	5.4763e-007 kg·m²	1.6034e-005 kg·m²	0.39459 kg·m²

Statistics			
Nodes	122	1806	6662
Elements	12	857	3460

Connections

TABLE 5: Model > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Contact On Update	Yes
Tolerance Type	Slider
Tolerance Slider	0.
Tolerance Value	2.0123e-003 m
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Same Body Grouping	Yes
Revolute Joints	Yes
Fixed Joints	Yes
Transparency	
Enabled	Yes

TABLE 6
Model > Connections > Contact Regions

Object Name	*Contact Region*	*Contact Region 2*	*Contact Region 3*	*Contact Region 4*	*Contact Region 5*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Contact	2 Faces				
Target	8 Faces				2 Faces
Contact Bodies	Part 1		Part 2		Part 3
Target Bodies	Part 3	Part 4	Part 3	Part 9	Part 4
Definition					
Type	Bonded				
Scope Mode	Automatic				
Behavior	Symmetric				
Suppressed	No				
Advanced					
Formulation	Pure Penalty				
Normal Stiffness	Program Controlled				
Update Stiffness	Never				
Thermal Conductance	Program Controlled				
Pinball Region	Program Controlled				

TABLE 7
Model > Connections > Contact Regions

Object Name	*Contact Region 6*	*Contact Region 7*	*Contact Region 8*	*Contact Region 9*	*Contact Region 10*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				

Contact	2 Faces	4 Faces	2 Faces	
Target	2 Faces		1 Face	8 Faces
Contact Bodies	Part 3	Part 5	Part 6	Part 7
Target Bodies	Part 9	Part 6	Part 8	
Definition				
Type	Bonded			
Scope Mode	Automatic			
Behavior	Symmetric			
Suppressed	No			
Advanced				
Formulation	Pure Penalty			
Normal Stiffness	Program Controlled			
Update Stiffness	Never			
Thermal Conductance	Program Controlled			
Pinball Region	Program Controlled			

TABLE 8
Model > Connections > Contact Regions

Object Name	*Contact Region 11*	*Contact Region 12*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Contact	2 Faces	
Target	8 Faces	2 Faces
Contact Bodies	Part 7	Part 8
Target Bodies	Part 9	
Definition		
Type	Bonded	

Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Thermal Conductance	Program Controlled
Pinball Region	Program Controlled

Mesh

TABLE 9
Model > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Advanced	
Relevance Center	Coarse
Element Size	Default
Shape Checking	Standard Mechanical
Solid Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Initial Size Seed	Active Assembly
Smoothing	Low

Transition	Fast
Statistics	
Nodes	15973
Elements	7650

TABLE 10
Model > Mesh > Mesh Controls

Object Name	*Body Sizing*
State	Fully Defined
Scope	
Scoping Method	Geometry Selection
Geometry	9 Bodies
Definition	
Suppressed	No
Type	Element Size
Element Size	2.e-002 m
Edge Behavior	Curve/Proximity Refinement

Static Structural

TABLE 11
Model > Analysis

Object Name	*Static Structural*
State	Fully Defined
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Options	
Reference Temp	22. °C

TABLE 12
Model > Static Structural > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Output Controls	
Calculate Stress	Yes
Calculate Strain	Yes
Calculate Results At	All Time Points
Analysis Data Management	
Solver Files Directory	E:\rafi\results with angle 18\uni-steel Simulation Files\Static Structural\
Future Analysis	None

Save ANSYS db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No

TABLE 13
Model > Static Structural > Loads

Object Name	*Fixed Support*	*Moment*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	8 Faces	2 Faces
Definition		
Type	Fixed Support	Moment
Suppressed	No	
Define By		Components
X Component		154. N·m (ramped)
Y Component		0. N·m (ramped)
Z Component		0. N·m (ramped)
Behavior		Deformable

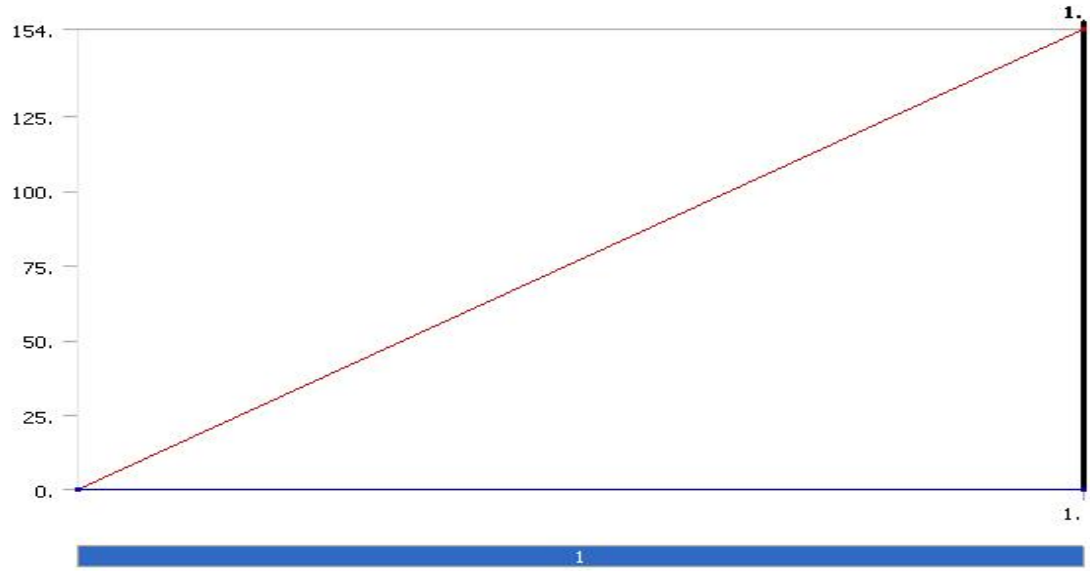


FIGURE 1
Model > Static Structural > Moment

Solution

TABLE 14
Model > Static Structural > Solution

Object Name	*Solution*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.

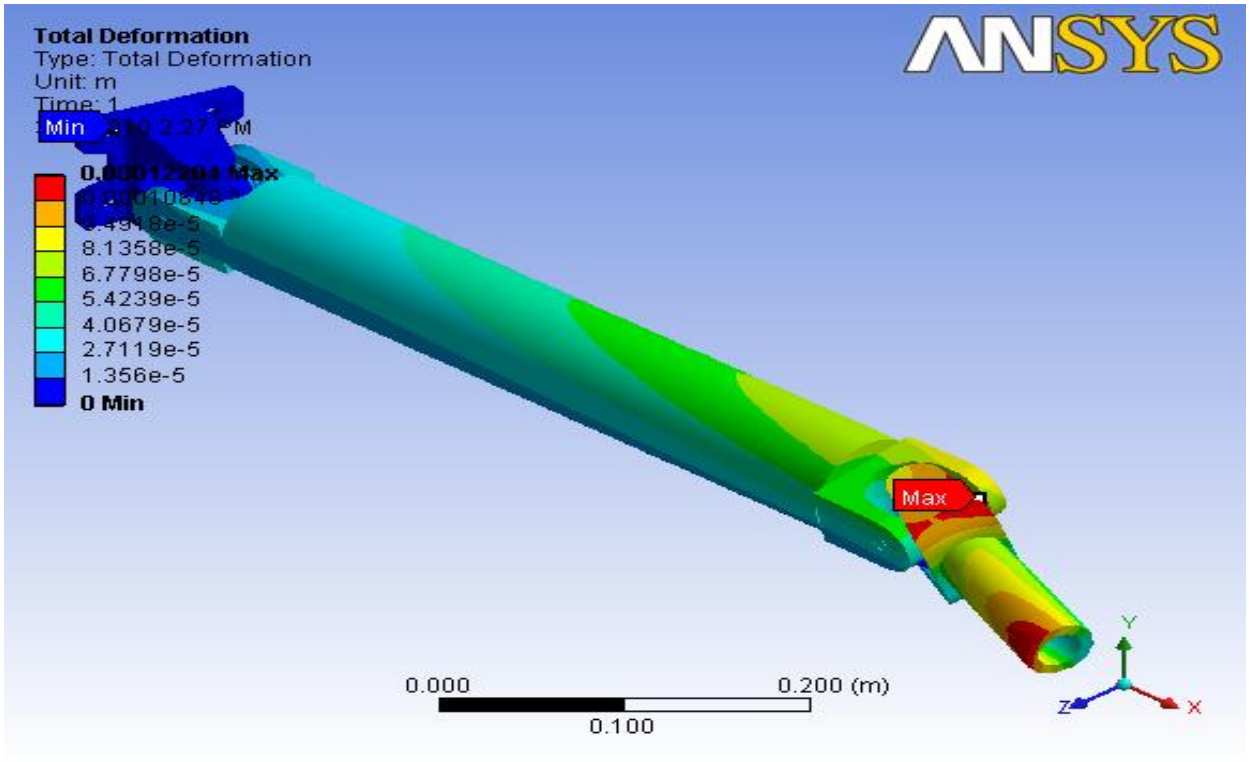
TABLE 15
Model > Static Structural > Solution > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All

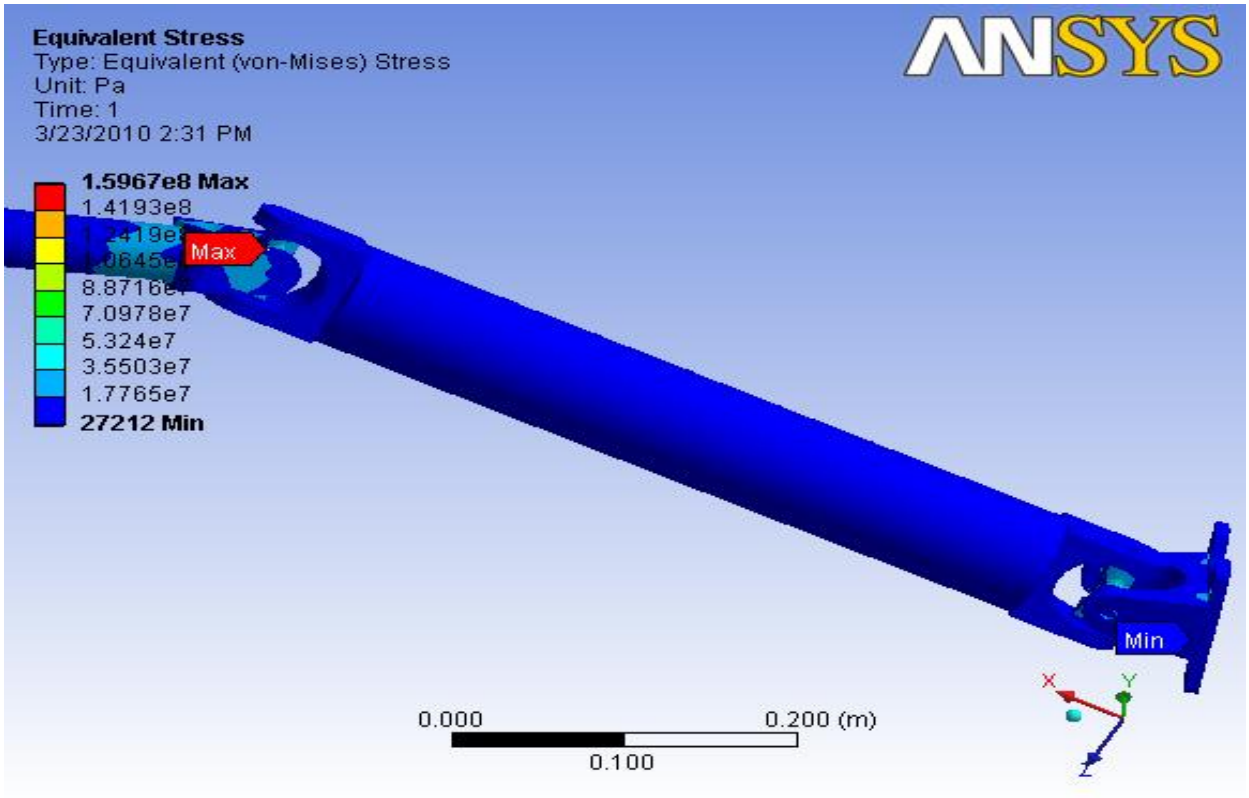
TABLE 16
Model > Static Structural > Solution > Results

Object Name	*Total Deformation*	*Equivalent Stress*	*Principal Stress*	*Shear Stress*
State	Solved			
Scope				
Geometry	All Bodies			
Definition				
Type	Total Deformation	Equivalent (von-Mises) Stress	Principal Stress	Shear Stress
Display Time	End Time			
Results				
Minimum	0. m	27212 Pa	-4.0827e+007 Pa	15620 Pa
Maximum	1.2204e-004 m	1.5967e+008 Pa	1.2355e+008 Pa	8.3394e+007 Pa
Minimum Occurs On	Universal joint		Spider(Slip yoke side)	Universal joint
Maximum Occurs On	Slip yoke	Spider(Slip yoke side)	Spider(universal joint side)	Spider(Slip yoke side)
Information				
Time	1. s			
Load Step	1			
Sub step	1			
Iteration Number	1			

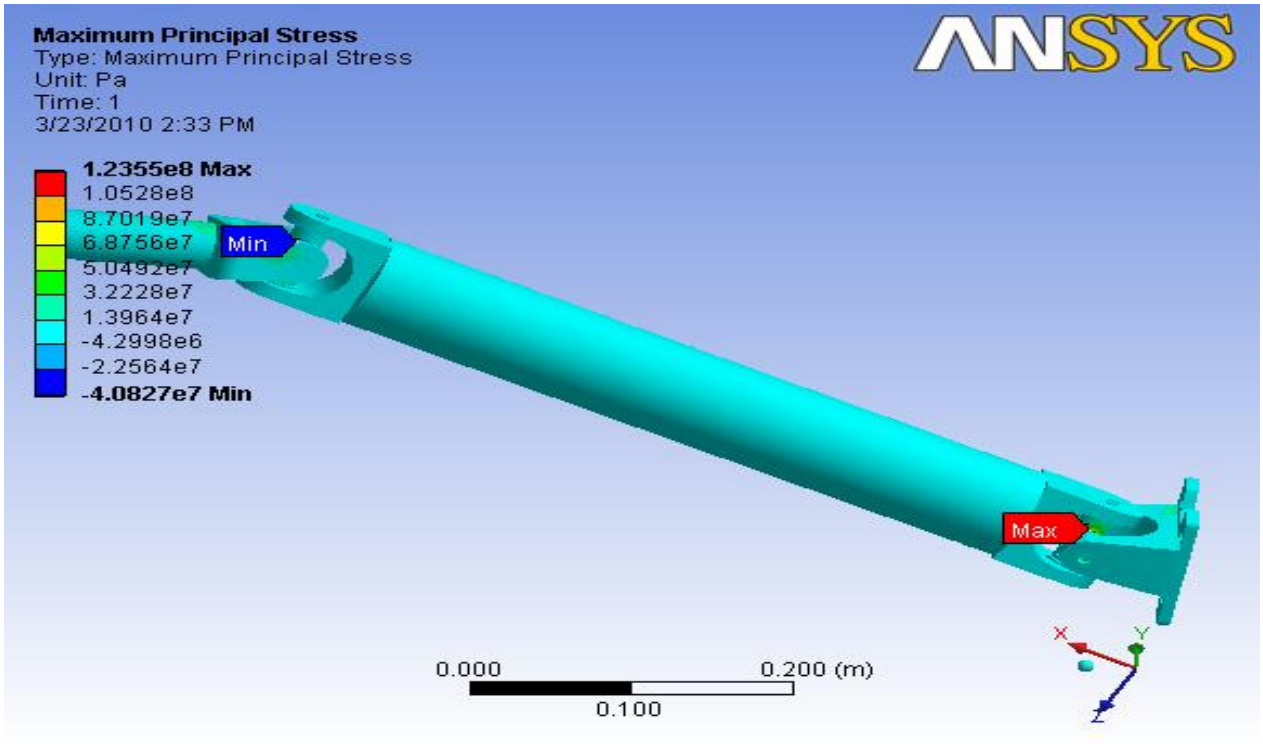
Total Deformation:



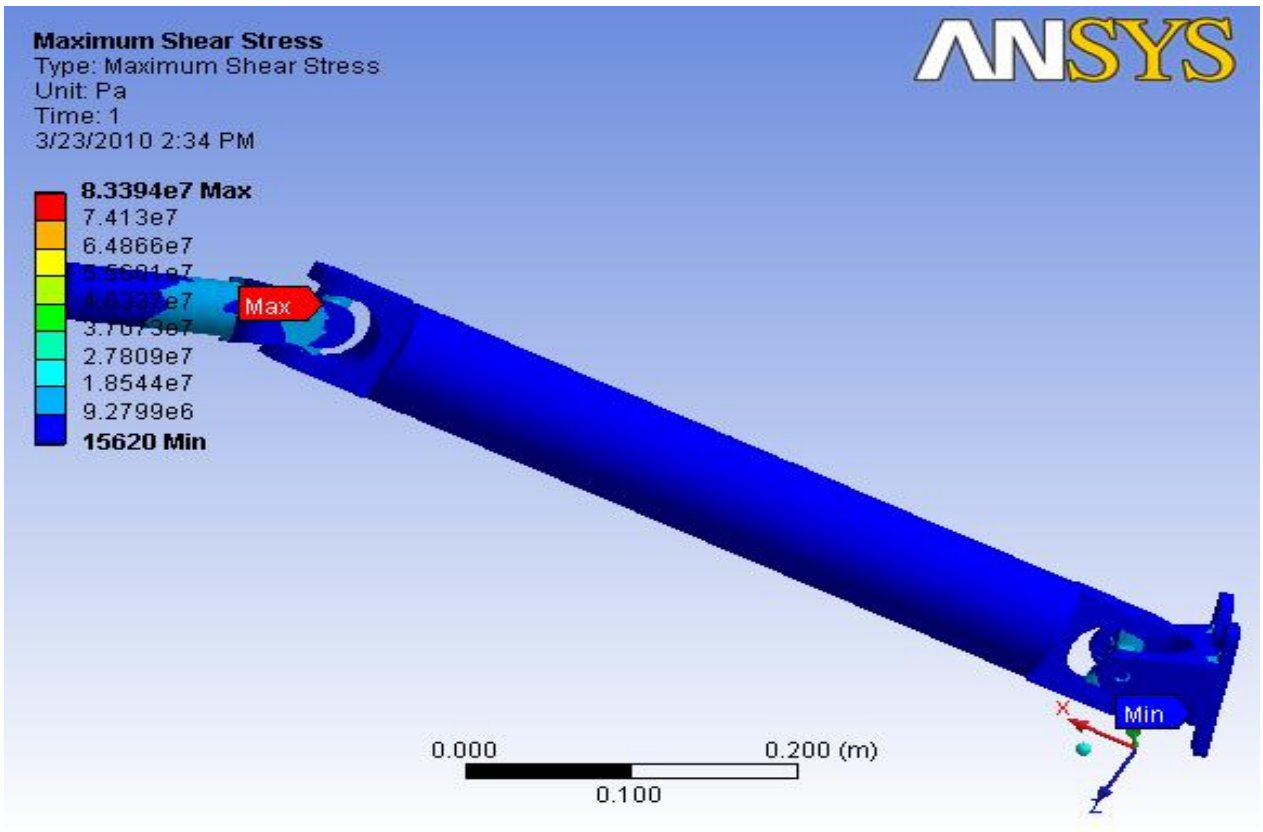
Equivalent Stress (von-Mises)



Maximum Principal Stress



Maximum Shear Stress



8.7 REPORT ON E GLASS

Project

	Tuesday, March 23, 2010
Last Saved	Tuesday, March 23, 2010
Product Version	11.0 Release

Material Data

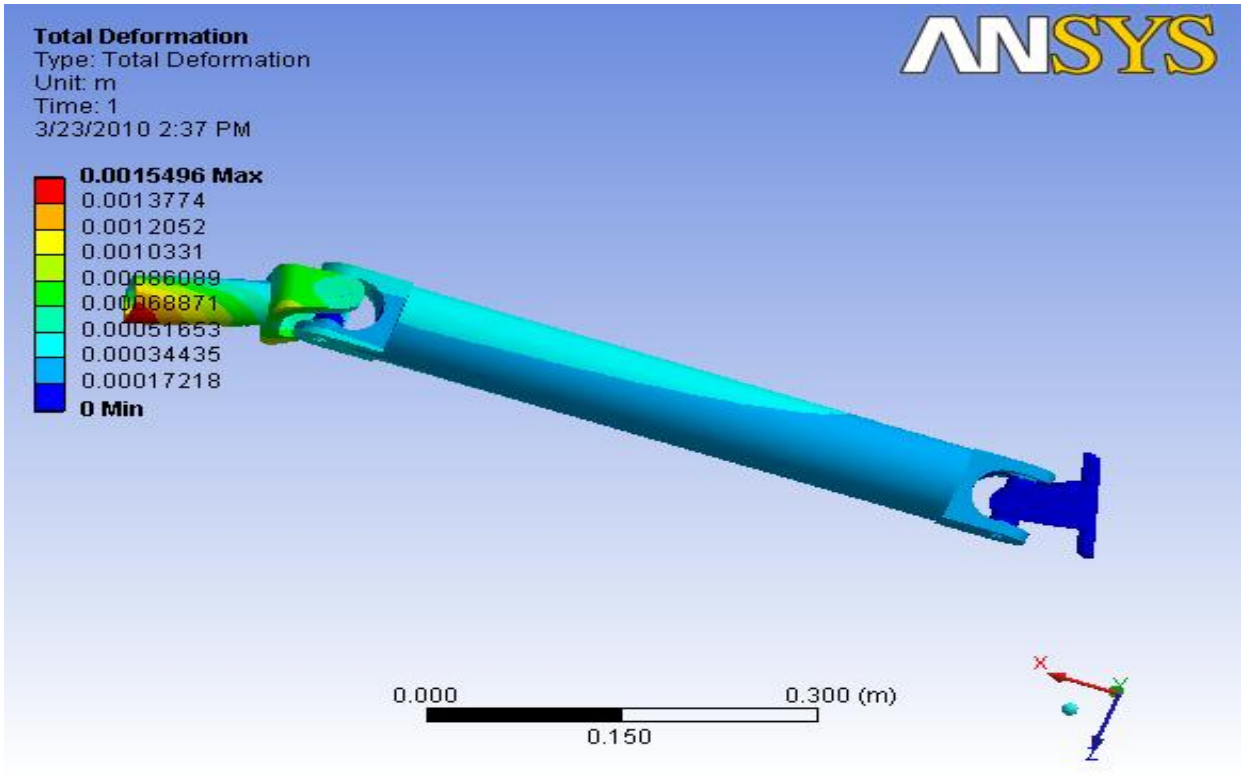
E-glass

TABLE 17
e-glass > Constants

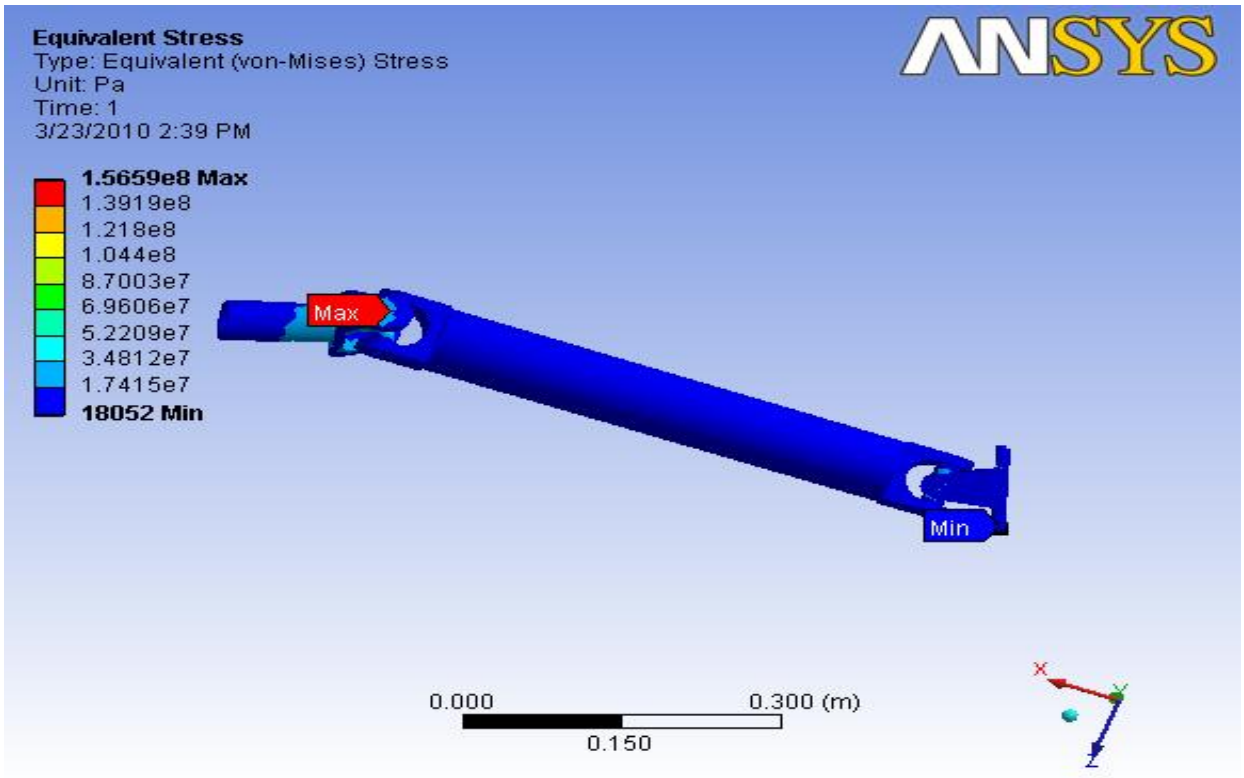
Structural	
Young's Modulus X direction	5.e+010 Pa
Young's Modulus Y direction	1.2e+010 Pa
Young's Modulus Z direction	1.2e+010 Pa
Major Poisson's Ratio XY	0.3
Major Poisson's Ratio YZ	0.3
Major Poisson's Ratio XZ	0.3
Shear Modulus XY	5.6e+009 Pa
Shear Modulus YZ	5.6e+009 Pa
Shear Modulus XZ	5.6e+009 Pa
Density	2000. kg/m³

RESULTS FOR E-GLASS				
Type	Total Deformation	Equivalent (von-Mises) Stress	Principal Stress	Shear Stress
Display Time	End Time		1. s	
Results				
Minimum	0. m	18052 Pa	-2.8742e+007 Pa	9859. Pa
Maximum	1.5496e-003 m	1.5659e+008 Pa	9.7387e+007 Pa	8.1624e+007 Pa
Minimum Occurs On	Universal joint		Spider(Slip yoke side)	Universal joint
Maximum Occurs On	Slip yoke	Spider(Slip yoke side)	Spider(universal joint side)	Spider(Slip yoke side)

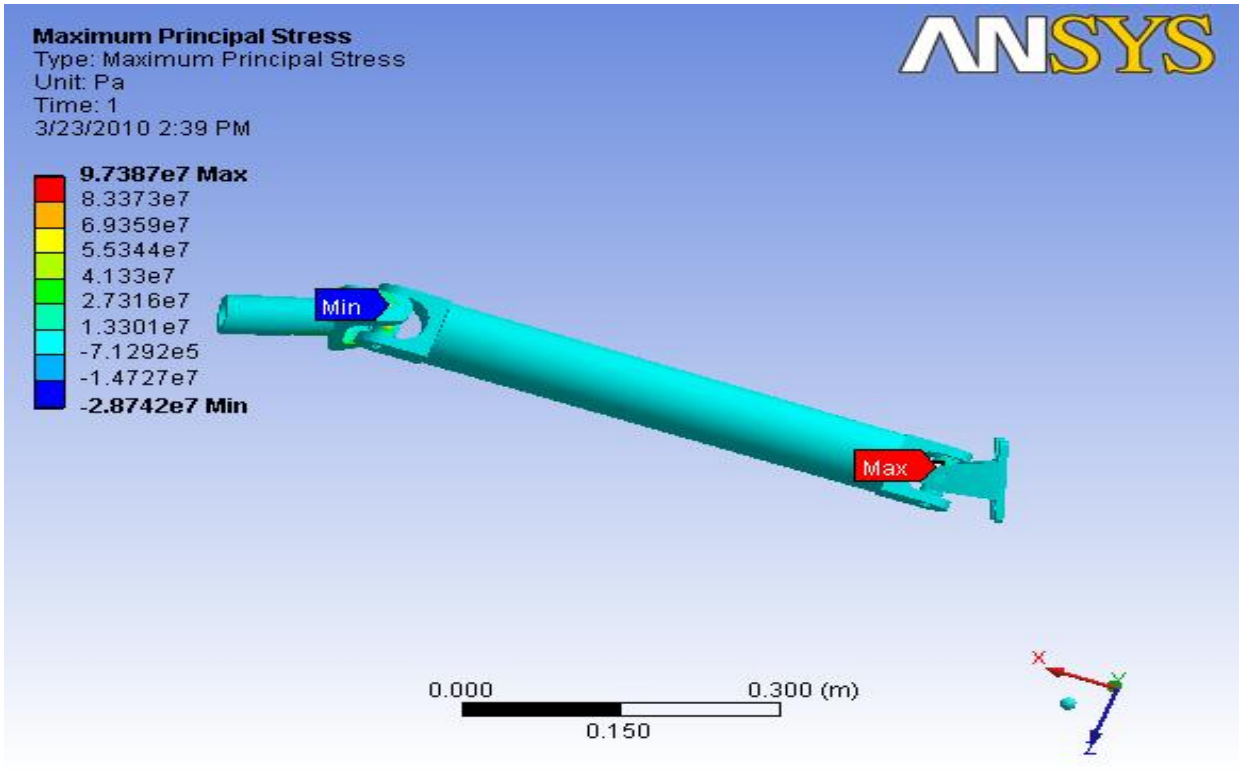
Total Deformation



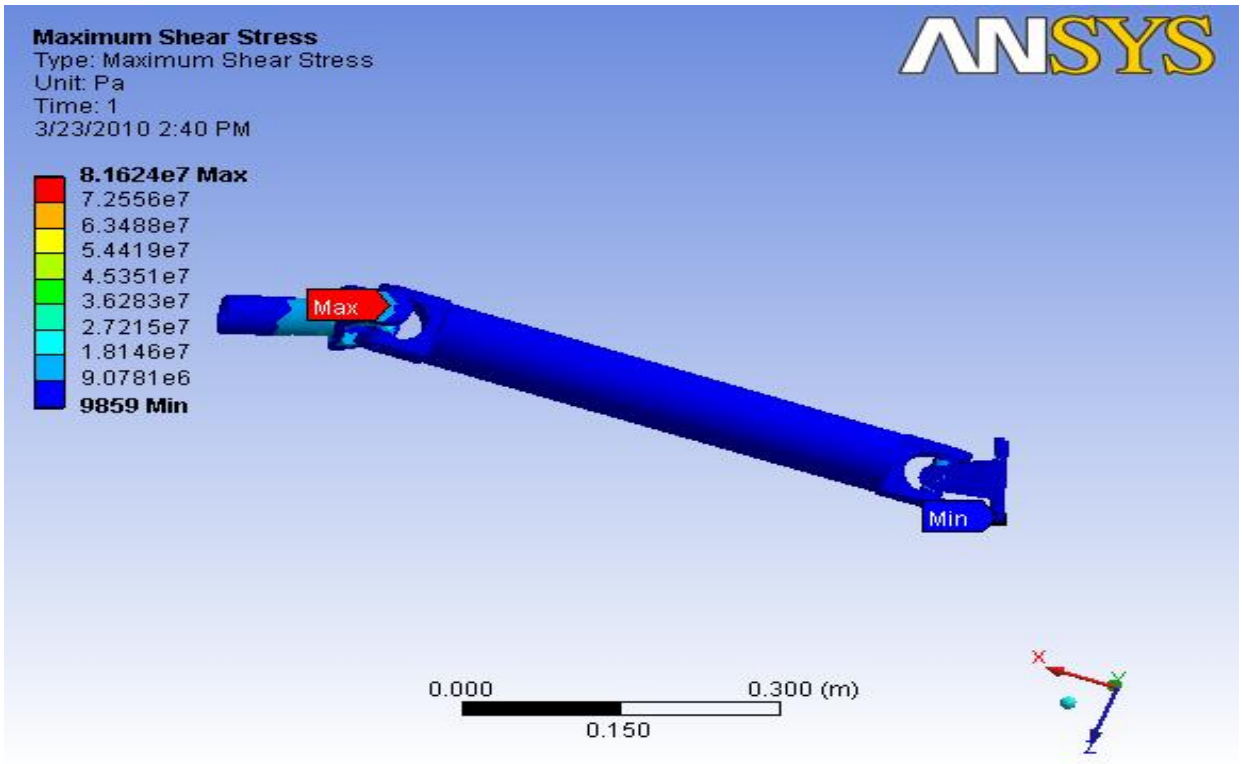
Equivalent Stress (Von-Mises)



Maximum Principal Stress



Maximum Shear Stress



8.8 REPORT ON E CARBON

Project

First Saved	Tuesday, March 23, 2010
Last Saved	Tuesday, March 23, 2010
Product Version	11.0 Release

Material Data

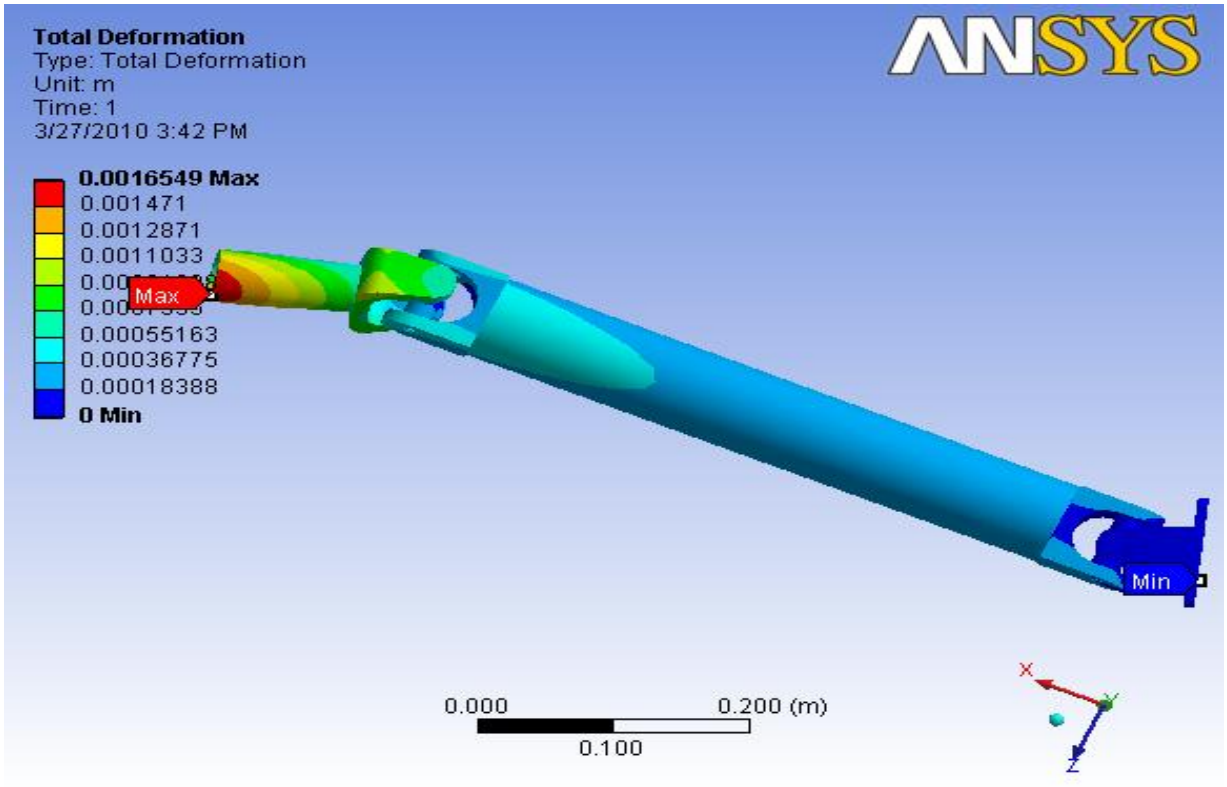
E-Carbon

TABLE 17
New Material > Constants

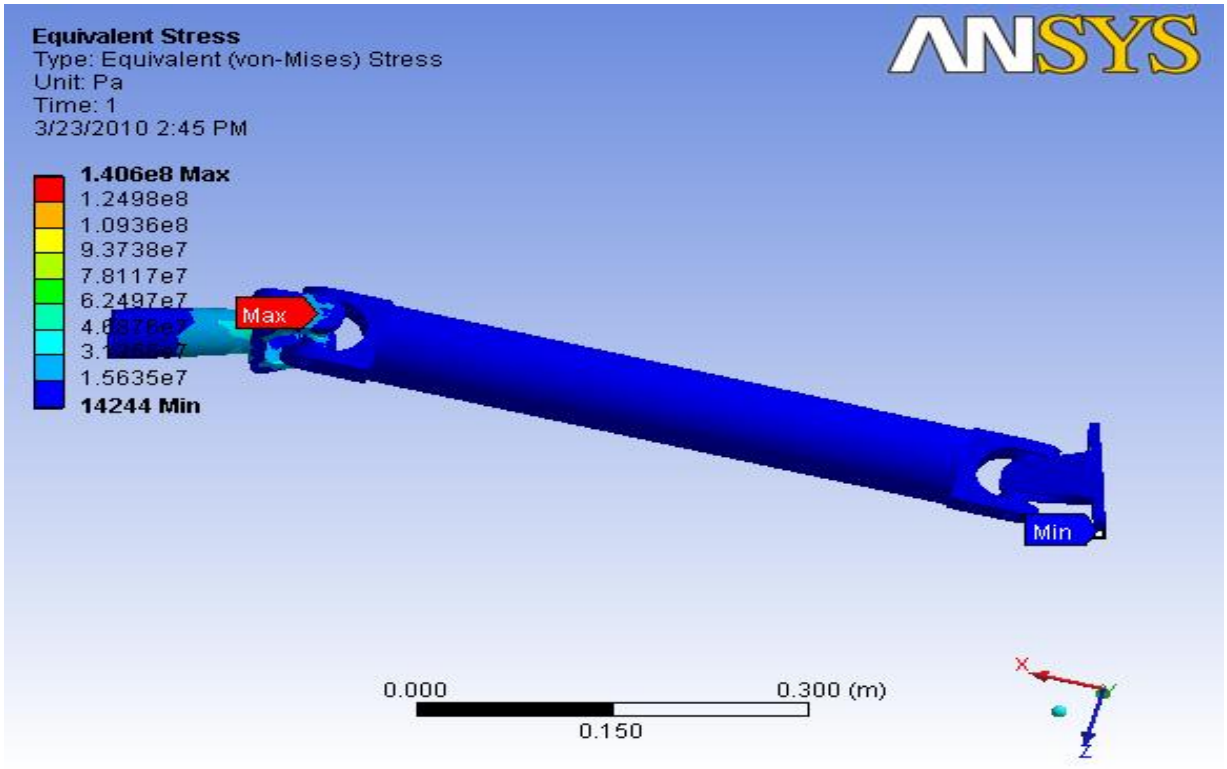
Structural	
Young's Modulus X direction	1.9e+011 Pa
Young's Modulus Y direction	7.7e+009 Pa
Young's Modulus Z direction	7.7e+009 Pa
Major Poisson's Ratio XY	0.3
Major Poisson's Ratio YZ	0.3
Major Poisson's Ratio XZ	0.3
Shear Modulus XY	4.2e+009 Pa
Shear Modulus YZ	4.2e+009 Pa
Shear Modulus XZ	4.2e+009 Pa
Density	1600. kg/m³

RESULTS FOR E-CARBON				
Type	Total Deformation	Equivalent (von-Mises) Stress	Principal Stress	Shear Stress
Display Time	End Time			
Results				
Minimum	0. m	0. Pa	-1.7418e+007 Pa	0. Pa
Maximum	1.6549e-003 m	1.4058e+008 Pa	1.1132e+008 Pa	7.3367e+007 Pa
Minimum Occurs On	Universal joint	Universal joint	Spider(Slip yoke side)	Universal joint
Maximum Occurs On	Slip yoke	Spider(Slip yoke side)	Spider(Universal joint side)	Spider(Slip yoke side)

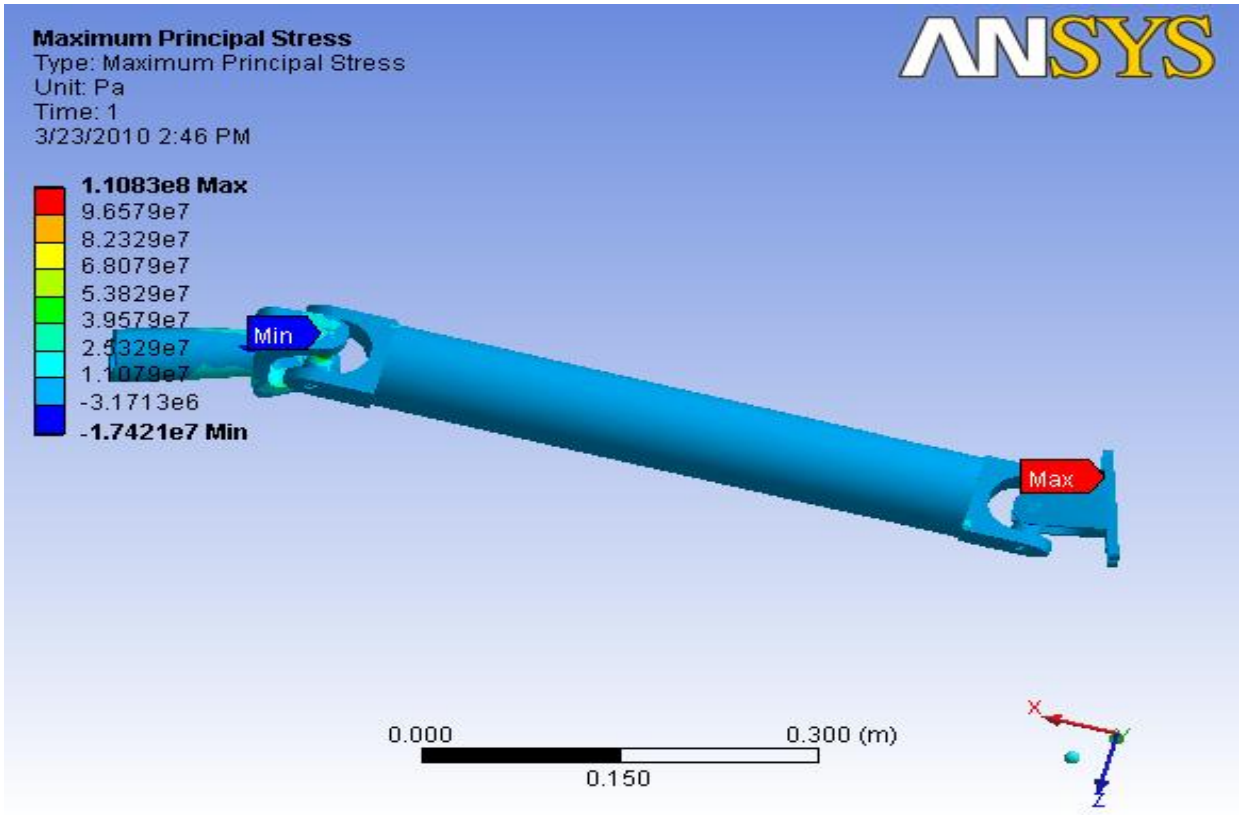
Total Deformation



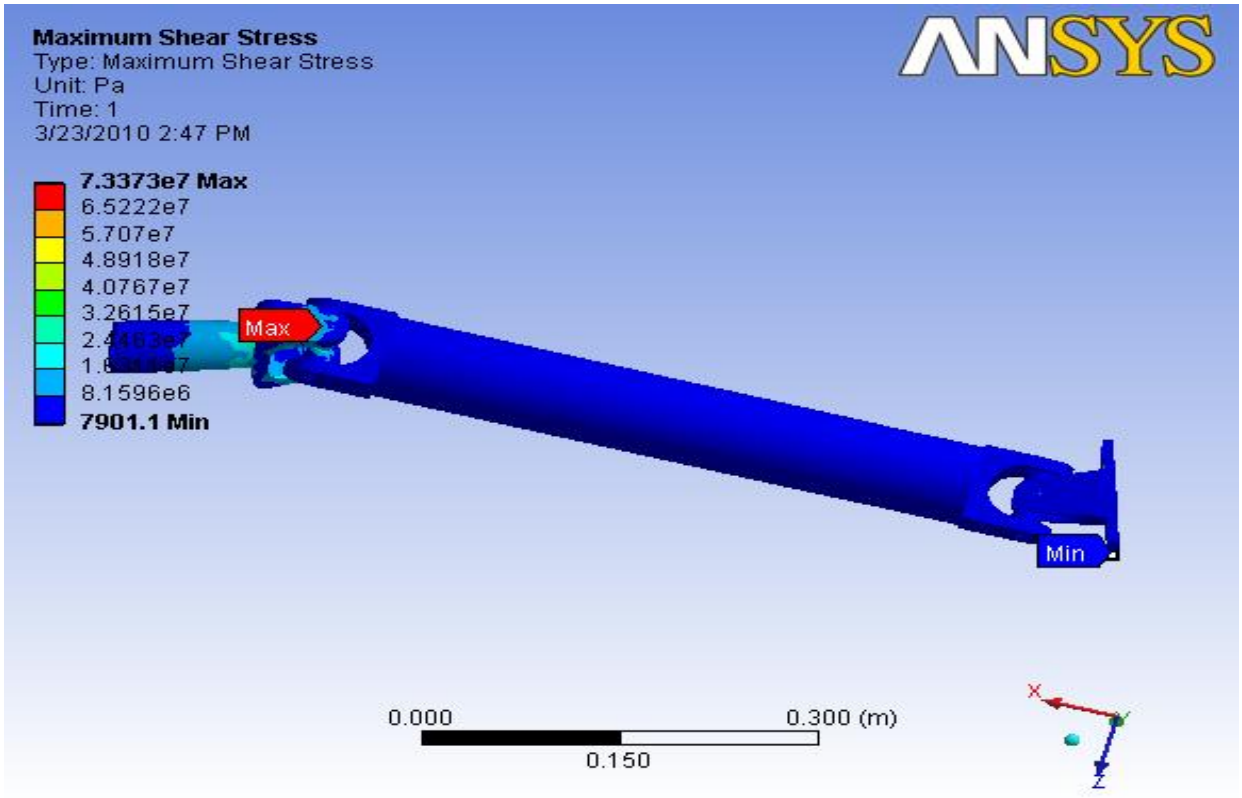
Equivalent Stress (Von-Mises)



Maximum Principal Stress



Maximum Shear Stress



8.9 REPORT ON E GLASS POLYESTER

Project

First Saved	Tuesday, March 23, 2010
Last Saved	Tuesday, March 23, 2010
Product Version	11.0 Release

Material Data

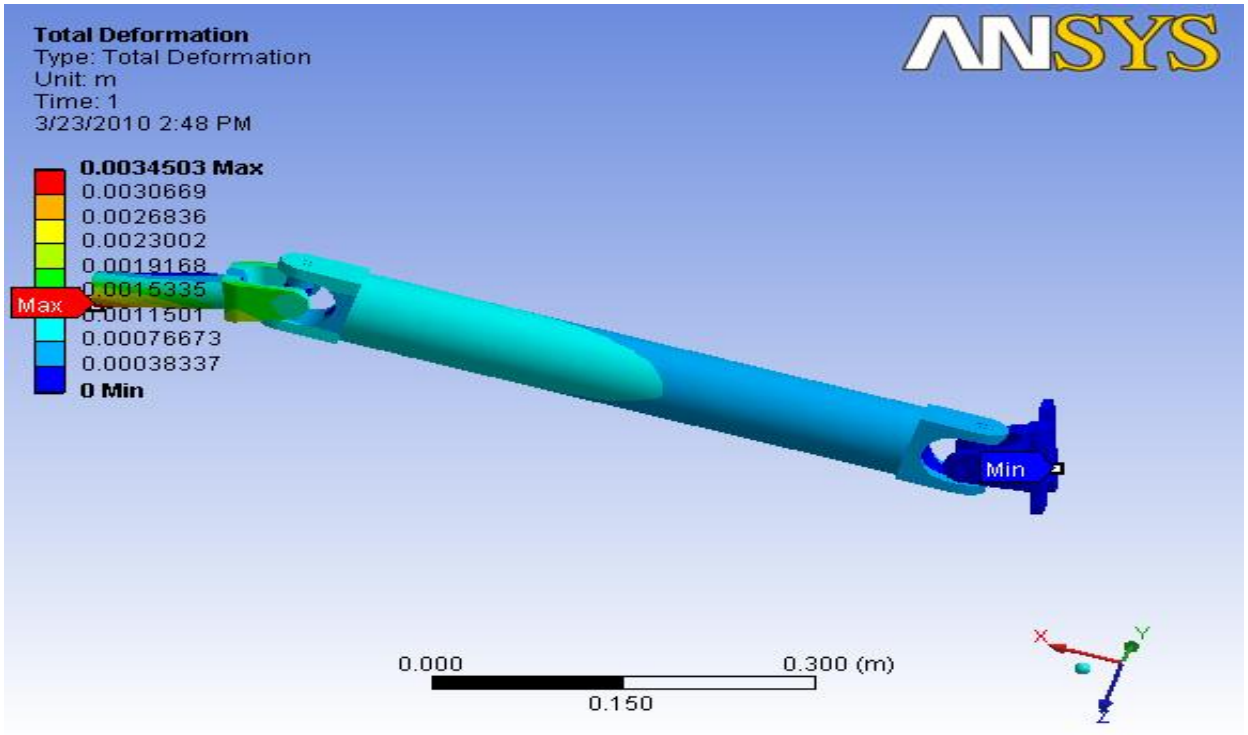
New Material 2

TABLE 17
New Material 2 > Constants

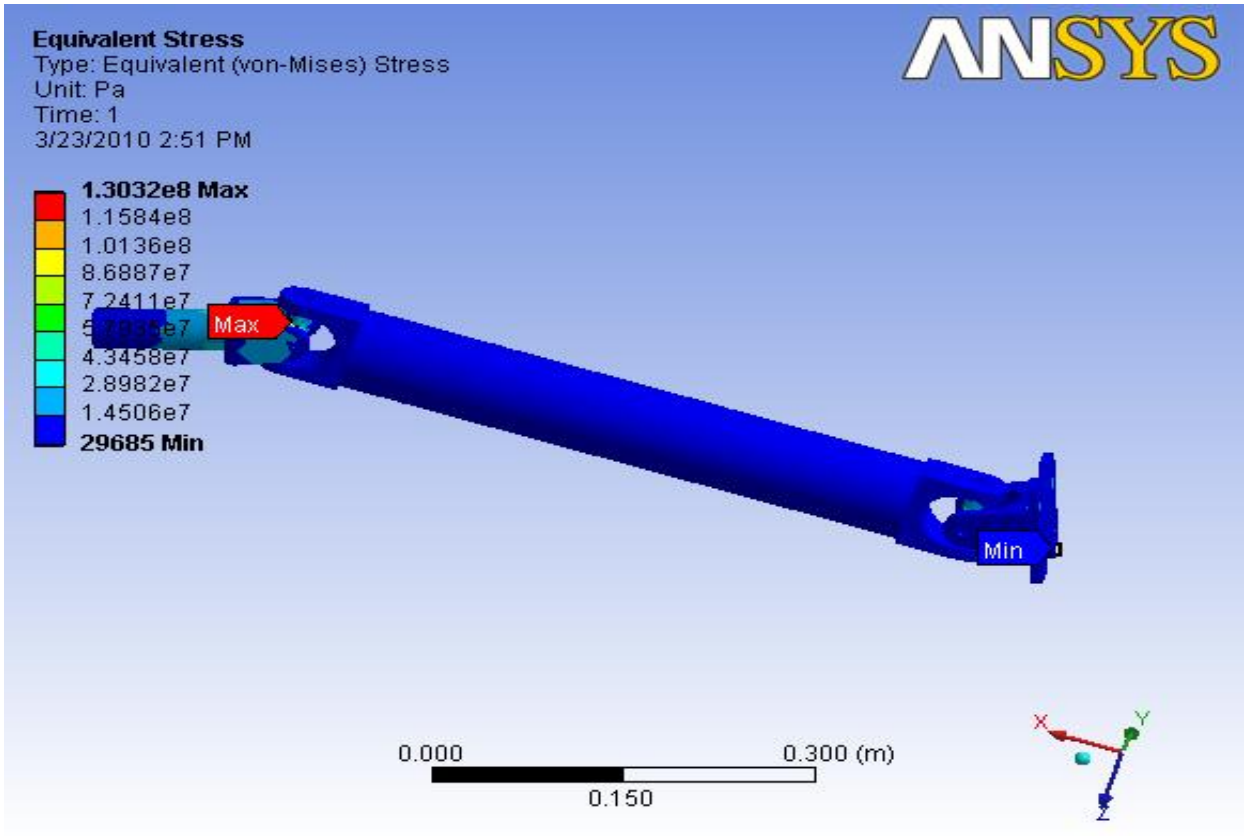
Structural	
Young's Modulus X direction	3.4e+010 Pa
Young's Modulus Y direction	6.53e+009 Pa
Young's Modulus Z direction	6.53e+009 Pa
Major Poisson's Ratio XY	0.217
Major Poisson's Ratio YZ	0.366
Major Poisson's Ratio XZ	0.217
Shear Modulus XY	2.433e+009 Pa
Shear Modulus YZ	1.698e+009 Pa
Shear Modulus XZ	2.433e+009 Pa
Density	2100. kg/m³

REPORT ON E-GLASS POLYESTER				
Type	Total Deformation	Principal Stress	Shear Stress	Equivalent (von-Mises) Stress
Display Time	End Time		1. s	
Results				
Minimum	0. m	-2.88e+007 Pa	17021 Pa	29685 Pa
Maximum	3.4503e-003 m	8.1044e+007 Pa	7.1391e+007 Pa	1.3032e+008 Pa
Minimum Occurs On	Universal joint	Spider(Slip yoke side)	Universal joint	
Maximum Occurs On	Slip yoke	Spider(Slip yoke side)	Spider(Slip yoke side)	

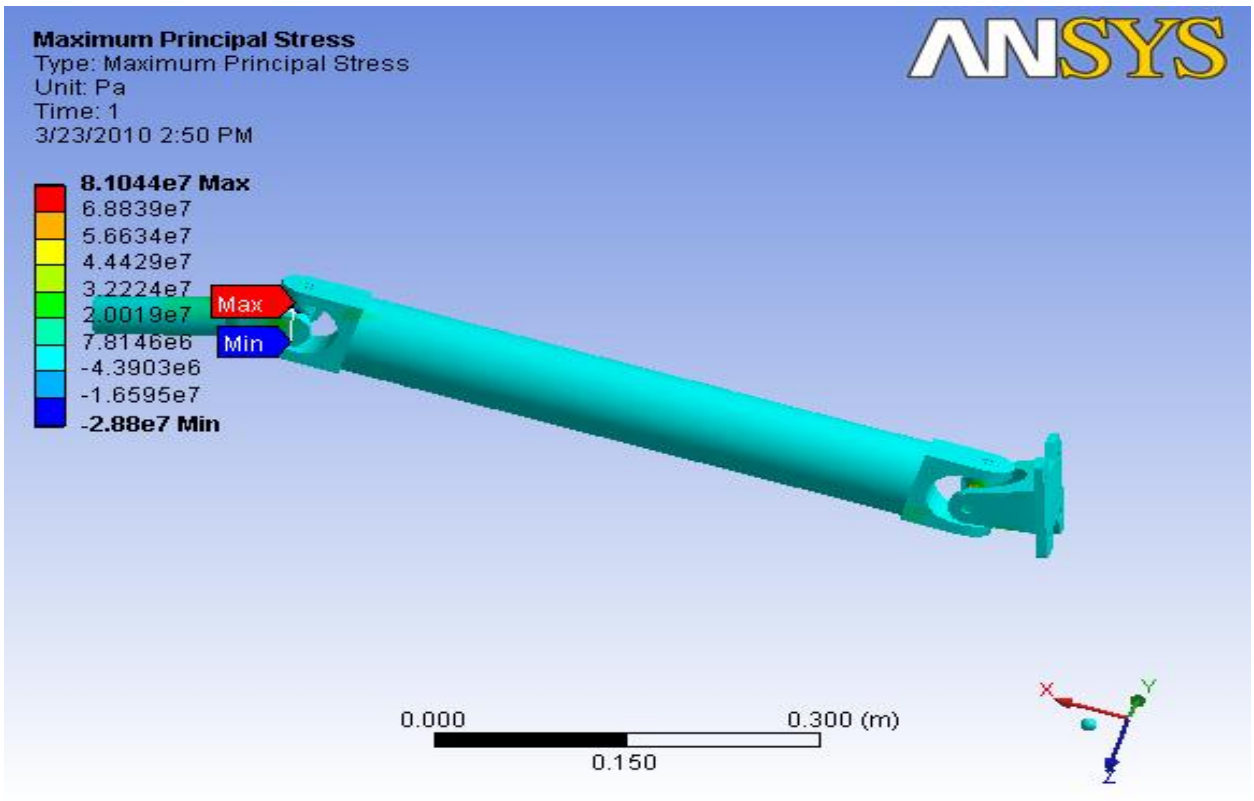
Total Deformation



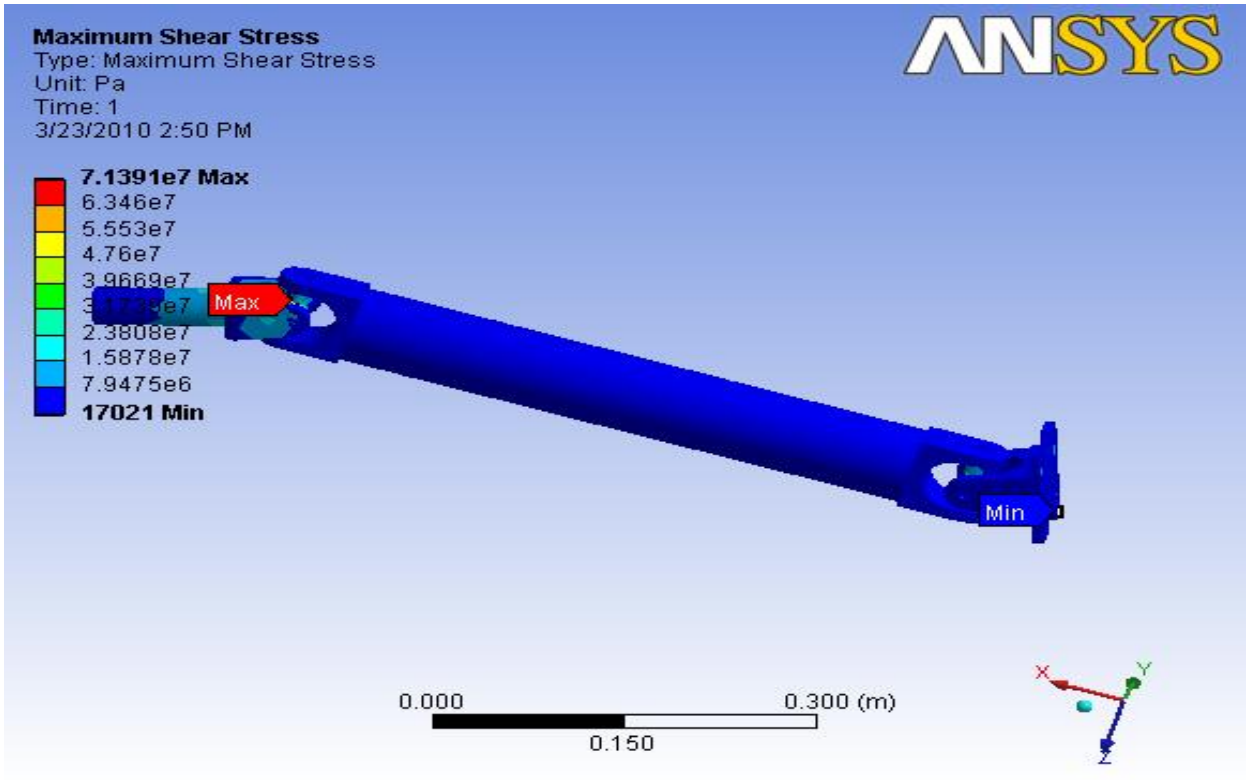
Equivalent Stress (Von-Mises)



Maximum Principal Stress



Maximum Shear Stress



RESULTS

PROPERTY/MATERIAL		STRUCTURAL STEEL	E-GLASS	E-CARBON	E-GLASS POLY.RESIN
WEIGHT(N)		177.2	46.6	37.3	49
%REDUCTION IN WE IGHT		-	74%	79%	72.4%
MAXIMUM PRINCIPAL STRESS(Pa)		1.28e008	9.74e007	1.11e008	8.11e007
%REDUCTION OF MAXIMUM PRINCIPAL STRESS		-	24%	13.3%	37%
MAXIMUM SHEAR STRESS(Pa)		8.34e007	8.16e007	7.34e007	7.14e007
% REDUCTION OF MAXIMUM SHEAR STRESS		-	2.2%	11.9%	14.4%
DEFORMATION (m)	MAXIMUM VALUE	0.00123	0.00155	0.00165	0.00345
	%INCREASE IN DEFORMATION	-	20.6%	24.5%	64.4%
EQUIVALENT STRESS OR VON-MISES STRESS (Pa) %	MAXIMUM VALUE	1.6e008	1.56e008	1.41e008	1.3e008
	ALLOWABLE STRESS	3.7e009	4e009	4.4e009	4.2e009
	%REDUCTION IN EQUIVALENT STERSS	-	2.5%	12%	19%

CHAPTER 9:
Implementation of Python Programming for Validation of Design and Finite Element Analysis Results of Automotive Composite Drive Shafts

9.1 Introduction

The design and analysis of automotive composite drive shafts involve analytical calculations, finite element simulations, and experimental validations. Traditionally, engineers rely on CAD software for modeling and ANSYS or Abaqus for finite element analysis. However, independent verification of simulation results is essential to ensure the reliability and accuracy of the design.

Python has emerged as a powerful open-source computational tool for engineering applications. Its extensive scientific libraries enable engineers to perform mathematical calculations, validate finite element results, automate repetitive computations, and optimize component designs. In the present work, Python programming is implemented to validate the analytical and finite element results obtained for the composite propeller shaft of the Toyota Qualis vehicle.

The validation process includes:

1. Torsional stress verification
2. Angle of twist calculation
3. Polar moment of inertia estimation
4. Weight prediction
5. Critical speed estimation
6. Factor of safety evaluation
7. Material performance comparison
8. Optimization of shaft dimensions

The Python-based validation framework provides an independent computational methodology to verify the structural performance of composite drive shafts.

9.2 Need for Python-Based Validation

Finite Element Analysis provides highly accurate results; however, validation is required because:

- Numerical errors may occur during meshing.
- Boundary condition assumptions may influence results.
- Material property assignment errors may occur.
- Mesh quality affects stress prediction.
- Computational verification improves confidence in results.

Python enables engineers to compare theoretical predictions with FEA outputs and quantify deviations.

9.3 Python Libraries Used

The following libraries are employed:

Library	Purpose
NumPy	Numerical computations
Pandas	Data handling and reporting
Matplotlib	Visualization
SciPy	Optimization
SymPy	Symbolic mathematics

Python Installation

```
pip install numpy
pip install pandas
pip install matplotlib
pip install scipy
pip install sympy
```

Import Libraries

```
import numpy as np
import pandas as pd
import matplotlib.pyplot as plt
from scipy.optimize import minimize
import sympy as sp
```

9.4 Validation of Torsional Stress

The maximum shear stress developed in the propeller shaft is calculated using classical torsion theory.

Governing Equation

$\tau = T r / J$

where

τ = Shear Stress (MPa)

T = Applied Torque (N-mm)

r = Outer Radius (mm)

J = Polar Moment of Inertia (mm^4)

Python Program

```
import numpy as np

Torque = 151000

Do = 75

Di = 65

r = Do/2

J = (np.pi/32)*(Do**4 - Di**4)

tau = (Torque*r)/J

print("Maximum Shear Stress =", tau,"MPa")
```

Interpretation

The calculated stress is compared with the ANSYS stress results. If the difference remains below 5%, the numerical model is considered validated.

9.5 Validation of Angle of Twist

The angle of twist is calculated using:

$\theta = TL/JG$

Where:

L = Shaft Length

G = Shear Modulus

Python Program

```
import numpy as np

T = 151000

L = 1250

G = 80000
```

```
Do = 75

Di = 65

J = (np.pi/32)*(Do**4 - Di**4)

theta = (T*L)/(J*G)

print(theta)
```

Validation Criterion

Compare calculated twist angle with ANSYS total deformation and rotational displacement.

9.6 Validation of Weight Reduction

Weight reduction is the primary objective of composite shaft design.

Python Program

```
import numpy as np

rho = 7850

Do = 0.075

Di = 0.065

Length = 1.25

Area = (np.pi/4)*(Do**2-Di**2)

Volume = Area*Length

Weight = rho*Volume

print(Weight)
```

Composite Weight Calculation

The density value is replaced by:

- E-Glass Epoxy = 2000 kg/m³
- Carbon Epoxy = 1600 kg/m³

- E-Glass Polyester = 2100 kg/m³

Weight savings are calculated using:

weight_reduction = ((steel_weight-composite_weight)/steel_weight)*100

9.7 Validation of Factor of Safety

The safety factor is calculated as:

FOS = Yield Strength / Working Stress

```
yield_strength = 250

working_stress = 9.8

FOS = yield_strength/working_stress

print(FOS)
```

The results are compared with ANSYS safety factor plots.

9.8 Validation of Critical Speed

The shaft should operate below its critical speed.

```
import numpy as np

g = 9.81

delta = 0.0004

Nc = (30/np.pi)*np.sqrt(g/delta)

print(Nc)
```

The computed value is compared with modal analysis results obtained through FEA.

9.9 Material Performance Validation

Python is used to compare different shaft materials.

```
import pandas as pd

data = {
"Material":["Steel","E-Glass","Carbon Epoxy","E-Glass Polyester"],
"Density":[7850,2000,1600,2100],
"Youngs_Modulus":[210,72,140,40]
```

```
}

df = pd.DataFrame(data)

print(df)
```

The generated table allows validation of material selection criteria.

9.10 Graphical Validation Using Matplotlib

```
import matplotlib.pyplot as plt

materials=["Steel","E-Glass","Carbon Epoxy","E-Glass Polyester"]

density=[7850,2000,1600,2100]

plt.bar(materials,density)

plt.xlabel("Materials")

plt.ylabel("Density")

plt.title("Density Comparison")

plt.show()
```

The graph visually confirms the weight advantage of composite materials.

9.11 Optimization Using Python

Optimization helps identify the optimum shaft thickness.

```
from scipy.optimize import minimize

def objective(x):
    return x[0]

result = minimize(objective,[5])

print(result.x)
```

Optimization constraints include:

- Torsional stress
- Buckling resistance
- Critical speed

- Weight minimization

9.12 Validation of FEA Results

The validation process consists of:

1. Analytical Calculation
2. Python Verification
3. ANSYS Simulation
4. Comparative Assessment

Percentage Error

analytical = 8.75

ansys = 8.92

error = abs(ansys-analytical)/analytical*100

print(error)

Acceptance Criterion:

Error ≤ 5%

9.13 Advantages of Python-Based Validation

- Open-source platform
- Fast numerical computations
- Reduced human error
- Automatic report generation
- Integration with ANSYS and CATIA
- Optimization capability
- Reproducible research methodology

9.14 Future Scope

Future work may include:

- PyMAPDL integration with ANSYS
- Machine learning-based failure prediction
- Digital twin development
- Real-time health monitoring
- Multi-objective optimization of composite laminates

10.CONCLUSION:

By conducting analysis on three different composite materials

- E-GLASS POLYSTER has 19%reduction in Von-Mises stress and 72.4%reduction in weight than Structural Steel. But it has 64% increase in deformation than Structural Steel.
- E-CARBON has 12%reduction in Von-Mises stress and 79%reduction in weight than Structural Steel. But it has 24.5% increase in deformation than Structural Steel.
- E-GLASS has 2.5%reduction in Von-Mises stress and 74%reduction in weight than Structural Steel. But it has 20.6% increase in deformation than Structural Steel.

By the obtained results it can be conclude that the stresses induced in all the materials are within their allowable limits. And it can also be observed that the materials which develop less von-mises stress exhibit a little more deformation. Though E-Glass Polyester Resin induces 19% less stresses compared to structural steel, considering the changes in both deformation and stress and weight (which is least - 1600 kg/m^3 among all the above materials), it can be concluded that E-CARBON can be used instead of conventional material like structural steel. So that the weight and stresses induced in the drive shaft can be considerably decreased.

Python programming provides an effective computational framework for validating analytical and finite element results of automotive composite drive shafts. The methodology developed in this study verifies torsional stress, deformation, critical speed, factor of safety, and weight reduction predictions. The validation confirms the accuracy of FEA results and establishes Python as a reliable tool for computational mechanics and design optimization of composite propeller shafts. The approach can be extended to advanced automotive components and next-generation lightweight vehicle structures.

References

- M. A. Badie, A. Mahdi, A. R. Abutalib, E. J. Abdullah and R. Yonus, International Journal of Engineering and Technology, Vol. 3, No.2, 2006, pp. 227-237. Mack, J., Advanced polymer composites, Mater.Edge, 18, January 1988.
- Schwartz, M.M., Composite Materials Hand Book, McGraw-Hill, New York, 1984.
- Ashby, M.F., On Engineering Properties of Materials, Acta Metall., 37, 1273, 1989.
- Jones, R.M., 1990,*Mechanics of Composite Materials*, 2e*,* McGraw-Hill Book Company, New York.
- Aurtar K.Kaw, 1997,*Mechanics of Composite Materials*, CRC Press,New York..
- Belingardi.G, Calderale.P.M. and Rosetto.M., 1990, "Design Of Composite Material Drive Shafts For Vehicular Applications",*Int.J.ofVehicle Design*, Vol.11,No.6,pp. 553-563.
- John W. Weeton et. al. 1986,"*Engineers guide to composite materials*, *American Society for Metal,* New York.
- Gill cooks composite resins for the 90s, M.C.Gill Doorway, 7, 27, spring 1990.
- Partridge, I.K., Advanced Composites, Elsevier Applied science, New York, 1989.
- Stephen R. Swanson, 1997, *Introduction to Design and Analysis with Advanced Composite Materials*, Prentice-Hall International, Inc.
- Dr.Kirpal Singh, Automobile Engineering, Vol. 1, 11th edition, 2008, Standard Publications Distributors, India.
- Automobile Engineering' of TATA Mc GRAW HILL by K.K.JAIN & R.B.ASTAHANA.

- JN Reddy, An Introduction to Finite Element Method, 8th edition, 2007, Me Graw Hill, India.
- CATIA Manual, Engineers Cadd Centre Pvt.Ltd.,
- http://en.wikipedia.org/wiki/ANSYS
- http://www.cybersteering.com/cbmain/utilcars/qualis_gs.html
- http://en.wikipedia.org/wiki/Composite_material
- http://en.wikipedia.org/wiki/CATIA

ABOUT THE BOOK

Automotive Composite Drive Shafts: Design, Computational Analysis, and Optimization Using Python provides a practical and research-oriented approach to the design and performance evaluation of modern composite drive shafts. The book integrates composite materials, mechanical design, computational analysis, finite element concepts, and Python programming to address real-world automotive engineering challenges.

Through detailed case studies, mathematical modeling, and Python-based simulations, readers learn how to analyze torsional behavior, evaluate material performance, optimize structural parameters, and develop lightweight, high-strength drivetrain components. The book serves as a valuable resource for students, researchers, academicians, and practicing engineers interested in advanced automotive design and computational engineering.

KEY FEATURES OF THE BOOK

- Composite drive shaft design and analysis
- Python-based engineering computations
- Material selection and optimization techniques
- Stress, deformation, and vibration analysis
- CAD and finite element modeling concepts
- Lightweight automotive component design
- Practical case studies and numerical examples
- Applications in automotive and mechanical engineering

Scan this QR Code &visit us:

Published by:
The Institute for Innovations in Engineering and Technology (IIET)
www.theiiet.com
contact@theiiet.com

ISBN 978-8-16-861440-6